Intentional Blank

LS-DYNA for Engineers

A Practical Tutorial Book

By Ryan Lee
Aerospace Engineer, CAE Expert, Failure Analyst

LS-DYNA for Engineers

A Practical Tutorial Book

First Edition 2019

Author Ryan Lee

Publisher BW Publications

Copyrightⓒ BW Publications 2019, All rights reserved

The contents of this publication are the sole intellectual properties of the authors. No part of this publication may be reproduced in any material form (including photocopying or storing in any medium by electronic means and whether or not transiently or incidentally to some other use of this publication) without the written permission of the copyright holder.
Homepage https://sites.google.com/view/bw-engineering/

ISBN 9781703208856

Author

Ryan Lee
University of Florida, Aerospace Engineering
Korea Advanced Institute of Science and Technology, Aerospace Engineering.
Currently working as an Aerospace engineer, Failure Analyst, CAE expert.

Contributing Author

Puneeth Latha
SJCE, Mechanical Department
Currently working as CAE engineer in Infosys Ltd.

Table of Contents

TABLE OF CONTENTS I

FOREWORD 3

PART A WHAT'S LS-DYNA? 5

Chapter 1. Introduction of LS-DYNA 6
Chapter 2. How to Use LS-PrePost 10
Chapter 3. Simple Impact Analysis: Ball-Plate Drop Analysis 21
Chapter 4. Element and Mesh Generation 34

PART B BEGINNER LEVEL 56

Chapter 5. Static Analysis of 3D Bracket 57
Chapter 6. Blade Impact Analysis of Composite Sandwich 67
Chapter 7. 3P Bending Analysis (Explicit vs. Implicit) 84
Chapter 8. Crush Box Analysis 104
Chapter 9. Penetration Analysis of Missile Projectile 116
Chapter 10. 2D Wing CFD Analysis with ICFD Solver 131

PART C EXPERIENCED LEVEL 144

Chapter 11. SPH Bird Strike Analysis 145
Chapter 12. Water Landing Analysis with ALE Method 163
Chapter 13. Explosive Analysis 181
Chapter 14. Random Vibration Analysis of Bogie Frame 195
Chapter 15. Ballistic Analysis of Woven Fabric 213
Chapter 16. Door Impact Analysis with Spot Weld Joint 229
Chapter 17. Floor Panel analysis with Restart 252

APPENDIX USEFUL TIPS AND KNOW-HOW 262

A1. Unit System 263
A2. Time Step in Explicit Scheme 267
A3. Contact Algorithms in LS-DYNA 271
A4. Parallel Analysis and Single/Double Precision 281
A5. Convergence Problems 284
A6. Useful Reference Site 289

EPILOGUE 291

Models used in the Workshops can be downloaded by scanning the QR code given in the Appendix.

Foreword

FEA (Finite element analysis) is widely used in solving engineering problems and has become an essential design tool for engineers. LS-DYNA is one of the best explicit finite element analysis software made by Livermore Software Technology Corporation (LSTC) which is used by engineers to predict the highly non-linear engineering problems. This book presents 17 chapters including workshop engineering problems and can serve as a self-learning guidebook that enables students and engineers in civil, mechanical, aerospace and biomedical engineering to solve practical finite-element based problems. This book guides readers by providing a step by step procedure to create the FE model using LS-Prepost. This book adopts a “hands-on, practice, workshop” experience, that is focused on providing practical implementing knowledge for FE analysis with LS-DYNA, rather than emphasizing mathematical theory. The materials in the book have been developed by the authors for the purpose of training post-graduate students and engineers. The reader will be guided from starting LS-DYNA modeling sessions to constructing parts and assembly, creating mesh, generating analysis keywords and reviewing results. The chapters in the book can be read sequentially, depending on the needs of the reader. Several examples, graduate-level and industrial application courses that can be thought, together with the corresponding chapters. The first chapter gives an introduction to LS-DYNA FE analysis software. From chapter 2 through 4, the book introduces the general work environment of LS-Prepost to create a model and solve analysis with a couple of examples. Part B introduces models from various fields so that users can gain experience in various fields of analysis through example lessons in LS-DYNA's strong analysis field. Part C introduces seven workshop models including industrial-FE models with meshless methods, ALE methods, explosive analysis, high-speed impact analysis, etc. Additionally, useful tips and know-how in FE analysis are summarized in the appendix. Fundamental approaches of this book are that the more FE models you create, the better readers learn the FEA and the program. It is intended to help readers establish a solid basis for FE problem-solving capability with LS-DYNA. Finally, I wish to thank reviewers for their valuable comments and suggestions. Of course, we take full responsibility for any mistakes in the book. We also welcome any suggestions from readers that could help us improve the book in the following edition. We welcome all those who would like to co-work on FEA-based engineering projects in the future.

October 2019
Ryan Lee

Intentional Blank

Part A.

What's LS-DYNA?

This chapter consists of a basic introduction for readers who are new to LS-DYNA and contents that can learn the application concept through simple example exercises. If you have used LS-DYNA before, you can start from Part B.

You can download the training model files by scanning the QR code given in the Appendix.

CHAPTER 1. INTRODUCTION OF LS-DYNA

LS-DYNA is a general-purpose nonlinear analysis program based on the finite element analysis technique. Recently, it is been widely used in various engineering fields which requires Multi-physics FE Analysis. Although there are not many users, they are used extensively in the automotive industry for collision/impact analysis. LS-DYNA is known to be a very powerful program for analyzing dynamic behavior and any general mechanics analysis problem which are solved using ABAQUS, ANSYS and NASTRAN can also be analyzed with the help of LS-DYNA. Therefore, the General-Purpose LS-DYNA FE package is widely used not only in automobile companies but also in aviation, construction, defense, manufacturing and medical fields all over the world.

LSTC
Livermore Software
Technology Corp.

LS-DYNA®

▲ CI of LS-DYNA Corp., LSTC(Livermore Software Technology Corporation)

LS-DYNA provides a wide variety of functions, but it is very powerful especially for solving complex nonlinear analysis based on explicit analysis algorithms, providing excellent analysis results in the collision, explosion, and machining fields of automobiles. In general, finite element analysis classes in graduate schools do not have easy access to LS-DYNA because they are assigned to work on programs such as ABAQUS or ANSYS. However, there are few departments that have no other choice than using LS-DYNA for actual design/analysis in branches such as automotive, aviation and shipbuilding. There are a number of laboratories in the Graduate Engineering School that use LS-DYNA for a variety of research projects. However, there are not many users, so it is not easy to learn by yourself.

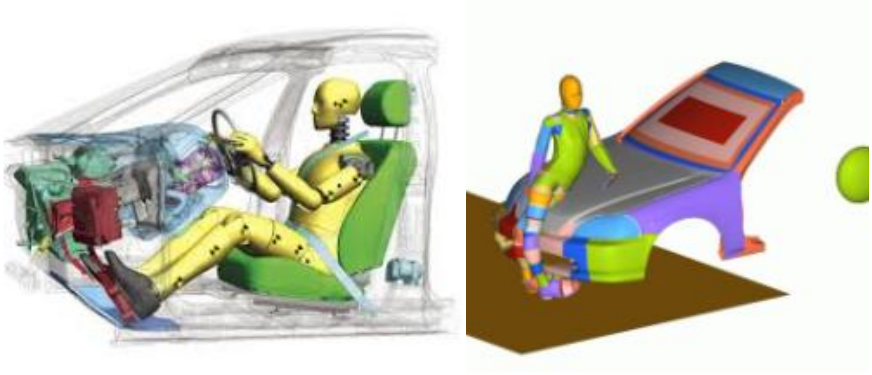
▲ Car crash analysis case (Dodge Neon vs. Toyota-RAV4)[1]

---

[1] Image courtesy: LSTC.com

Looking back at the history of LS-DYNA, it is almost in line with the period when FEM was very active. In 1976, Dr. John O. Hallquist, who was working at LLNL (Lawrence Livermore National Laboratory) in California, began developing the first 3D FEA program called DYNA3D. Interpretation codes were developed for explosives research conducted at LLNL, which was run by the US government. It was developed to analyze explosions of various bombs for weapons development. It was a time when the initial computer capabilities were not adequate and there was no 3D analysis program available commercially. The Early DYNA3D program was used in a variety of analyzes, in particular, it was carried out on the development of any weapons prior to the physical testing using explicit time integration (extrapolation).

DYNA3D made a great improvement in the 1980s with the rapid growth of computer capability. As the program became commercially available to the general public, many functions were added on request from several users. Various elements, contact, and material models have been added with numerous analysis algorithms improvement to enable a coupled analysis of the various physical problem. Structure-related interaction analysis, such as explosive structure or soil structure, has been made possible and the high-speed impact analysis has been significantly improved. In 1988, it was developed into a commercial company called LSTC. Since then, the program was renamed to LS-DYNA and many functions have been added with numbers of users gradually expanding across the world. Recently, the range of applications has been gradually expanded for user convenience and excellent analysis functions are added for general use including other analysis programs. LSTC is currently based in California and holds annual user conferences that are widely applied and used in various fields of research and have published a number of research papers.



▲ Driver / Pedestrian Crash Analysis [2]

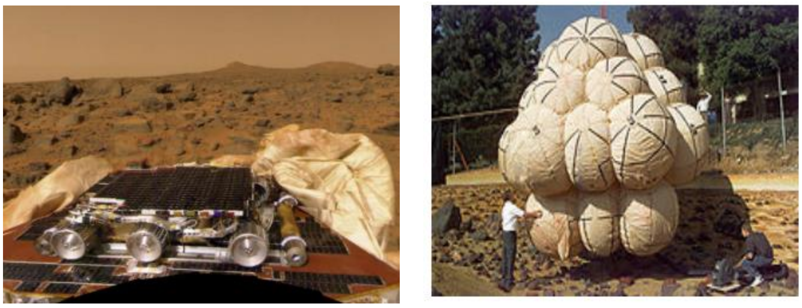
---

[2] Image courtesy : Center for applied biomechanics, University of Virginia, 2015

LS-DYNA Application

LS-DYNA is suitable for analysis of dynamic behavior mechanics involved with large deformation and it's relatively easy to solve due to the characteristics of the program, complex non-linear materials, and complex contact algorithm. It is possible to select not only explicit but also implicit analysis and it can be chosen according to the condition. It also includes functions such as Smooth Particle Hydrodynamics (SPH), Arbitrary Lagrangian-Eulerian (ALE) and Boundary Element Method (BEM) for solving multi-physics analysis. However, It can be identified that keyword-based modeling is somewhat inferior in user convenience compared to other large analysis programs. Also, the learning curve for understanding and using the keyword edition is inferior while resolving any analysis for the first time. In order to overcome this drawback, the pre-processing / post-processing program called LS-Prepost(GUI) is distributed for free online for LS-DYNA users.

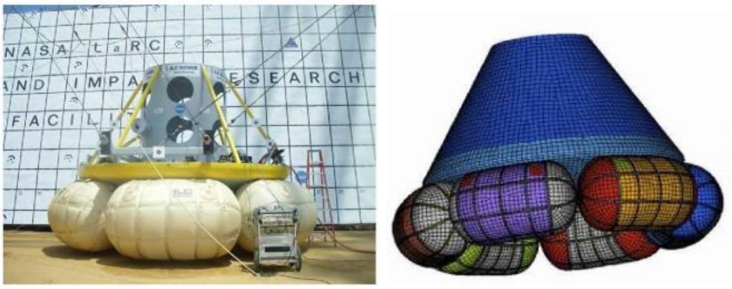
One of the advantages of LS-DYNA is that it can solve very high nonlinear multiphysics problems with a single code and also can be applied for various fields. It aims to be a "one code strategy" and provides a solution related to highly non-linear multiphysics (Multi-Physics + Multi-stage) where various physics are intertwined. Other programs often analyze multiple physics by binding and coupling codes from various sources. DYNA can be solved with a single code, which is quite efficient in the modeling process. For example, temperature-displacement or fluid-structure interaction analysis is a typical illustration containing multi-physics analysis. In addition, crash, airbag, seat belts and sensor analysis, which are specialized in the automotive industry, are also solved using LS-DYNA. A representative case of LS-DYNA is a landing simulation using Mars Pathfinder's Airbag, one of NASA's Mars exploration projects in the late '90s. Recently, the Orion Crew Module, a crew member of the space shuttle, has been successfully used in the airbag landing analysis.



▲ Airbag assisted the landing of NASA JPL Mars Pathfinder[3]

---

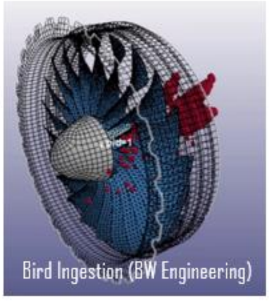
[3] Image courtesy of NASA JPL – Caltech, July 4, 1997

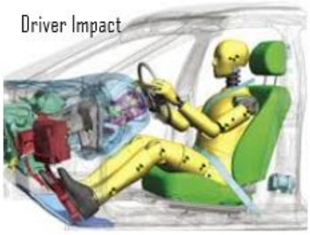


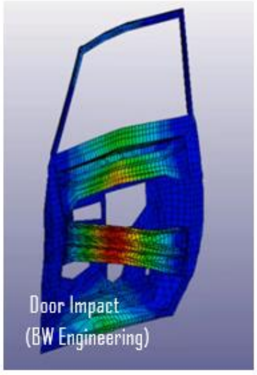
▲ Airbag Landing Test and Corresponding LS-DYNA model[4]

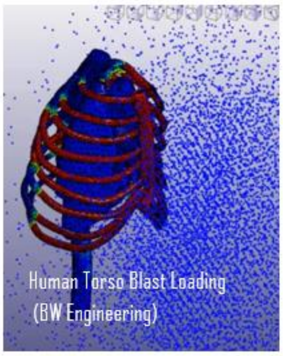
It is also widely used in most metal processing and molding processes analysis fields such as metal forming, stamping, rolling, extrusion, and machining. In addition, it is used in the fields of biomechanics such as biostructure analysis, implant analysis, seismic analysis such as earthquake analysis of buildings and building safety. The National Transport and Safety Board (NTSB) and the Federal Aviation Administration (FAA) in the US also use LS-DYNA as the standard analysis program in various certification tests and piloting procedures. The following figures show examples of representative analysis fields used in LS-DYNA analysis.

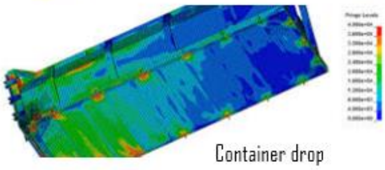
- ✓ **Crashworthiness/ Driver Impact/ Drop test simulation**
- ✓ **Mesh Free Method: ALE, EFG, SPH, Airbag particle**
- ✓ **Heat Transfer Analysis**
- ✓ **Metal Forming Analysis**
- ✓ **Earthquake Engineering**
- ✓ **Acoustic/Vibration/Fatigue**
- ✓ **Discrete element method**
- ✓ **CFD (incompressible, compressible)**
- ✓ **EM (Electromagnetism)**











---

[4] Image courtesy of R.B. Timmers et al, Modeling and Simulation of 2nd Generation Orion Crew Module Airbag Landing System, AIAA, 2009

Chapter 2. How to Use LS-Prepost

About LS-Prepost

What do we need to simulate any analysis using LS-DYNA? All we need is the LS-DYNA Solver, LS-Prepost and the analyst's brain. Here, LS-Prepost is a dedicated pre- and post-processing program for LS-DYNA. LSTC always distributes the latest version free of charge and it is an essential program to use LS-DYNA, so download the latest version and use it for practice. Please note that both Windows and Linux versions are available. As of October 19, version 4.6 is released, so download and install the latest version from the below link:

http://ftp.lstc.com/anonymous/outgoing/lsprepost/

When you run this program for the first time, you may feel that the GUI is very old. In fact, the users may feel it's not a very good program in terms of GUI. However, if you look closely at the features provided, you will often find features that are more superior to those offered by large companies and they offer quite a lot of features. If you are a beginner, it may take some time to learn, but once you get it right, it's a versatile program. In fact, many programs such as Hyperworks, Ansys and Patran can generate Dyna keywords, but LS-Prepost provided by the same company is the most compatible, so you must learn and become familiar with the LS-Prepost working environment.

All keywords provided by LS-DYNA can be created in LS-Prepost and as the name implies, both pre-process and post-process functions are included.

LS-Prepost
Preprocessor

- Generate Keyword
- Geometry
- Material
- Process

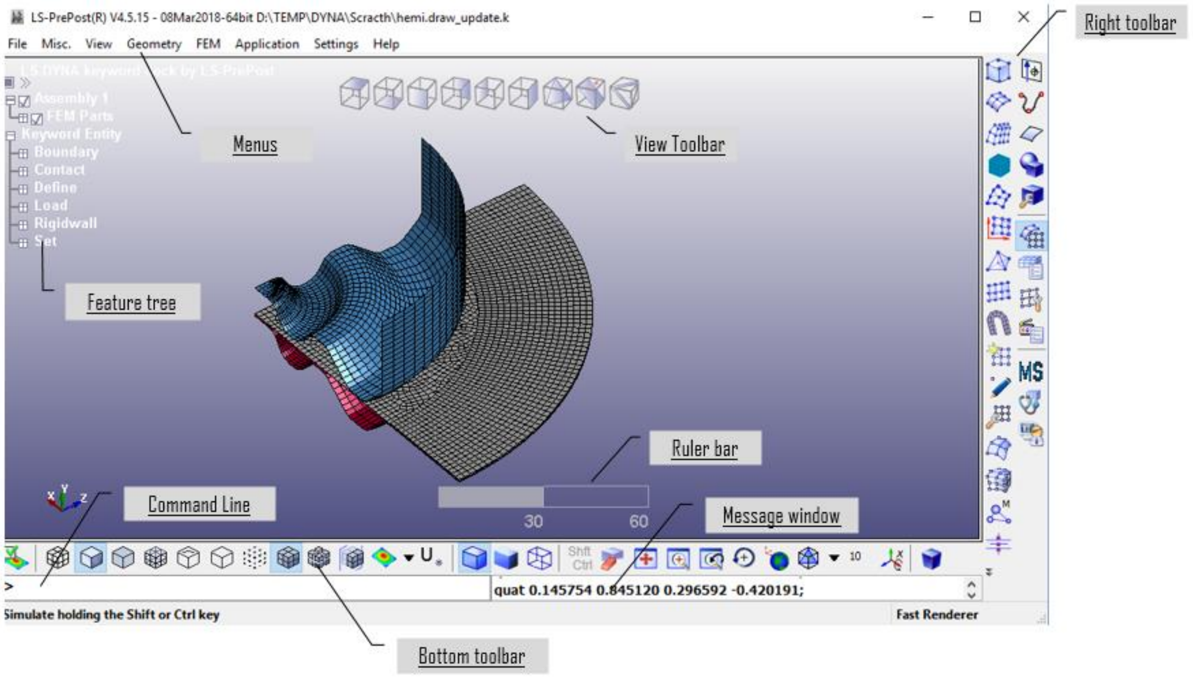
LS-DYNA
Solver

- Solve numerical model

LS-Prepost
Post-Processor

- View graphical results
- Plotting
- Extract numerical data

Let's run LS-Prepost first and look at the basic GUI. In this book, we have utilized version 4.5 released on May 18. The basic GUI has all the functions of a general preprocessor and does not differ much from other programs. The model feature tree in the viewport is very useful when modeling because all the components are shown in a tree structure.



LS-PrePost(R) V4.5.15 - 08Mar2018-64bit D:\TEMP\DYNA\Scracth\hemi.draw_update.k

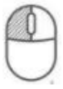
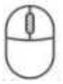
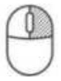
quat 0.145754 0.845120 0.296592 -0.420191;

Simulate holding the Shift or Ctrl key

▲ LS-DYNA's GUI screen

Like all other CAD/CAE systems, the ease of use of the mouse is a very important factor in successful modeling. Learn the following features from your hand tom familiarize the options:

Model manipulation

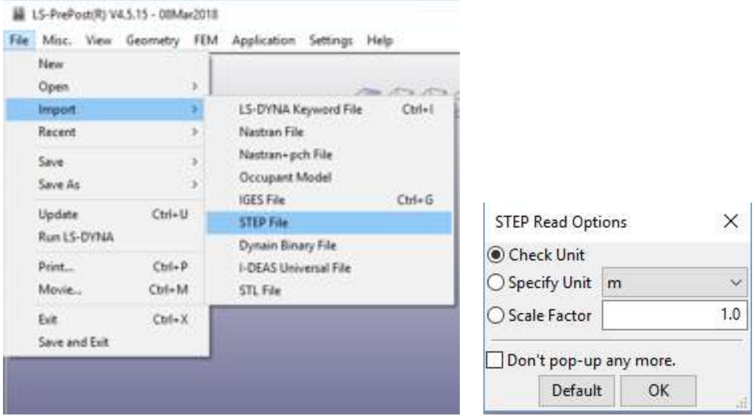
- **Shift +**  **→ Rotate**
- **Shift +**  **→ Translate**
- **Shift +**  **(or scroll wheel) → Zoom**

Graphic Selection

- **Select (single) → Left Click**
- **Select (Rectangle) → Left Click +Drag**
- **Select (Polygon) → Left Click + Left click → Right click at final**
- **Select (list) → Left Click + Drag and additional pick (Control+Left click)**

** If you press Control instead of Shift, only the line will be shown when you move the model.*

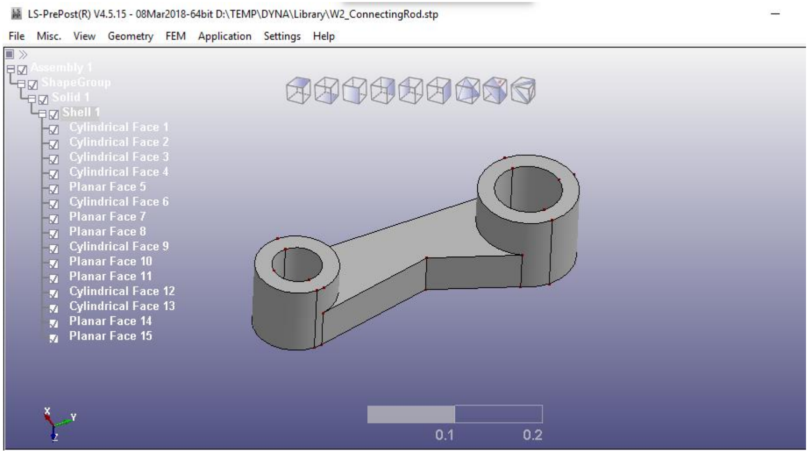
Now let's take a look at some basic menus. Select the File **Menu → Import → STEP** file and import the ***W2_connectingrod.stp*** model. When the import option floating/ popup window appears, select unit as m [meter] because the file is modeled in the metric unit. The File menu is a basic file management menu, so no further explanation is needed. The Run LS-DYNA function allows you to perform analysis directly by controlling the solver directly, which is further explained later.



● Check Unit
○ Specify Unit m
○ Scale Factor 1.0
☐ Don't pop-up any more.

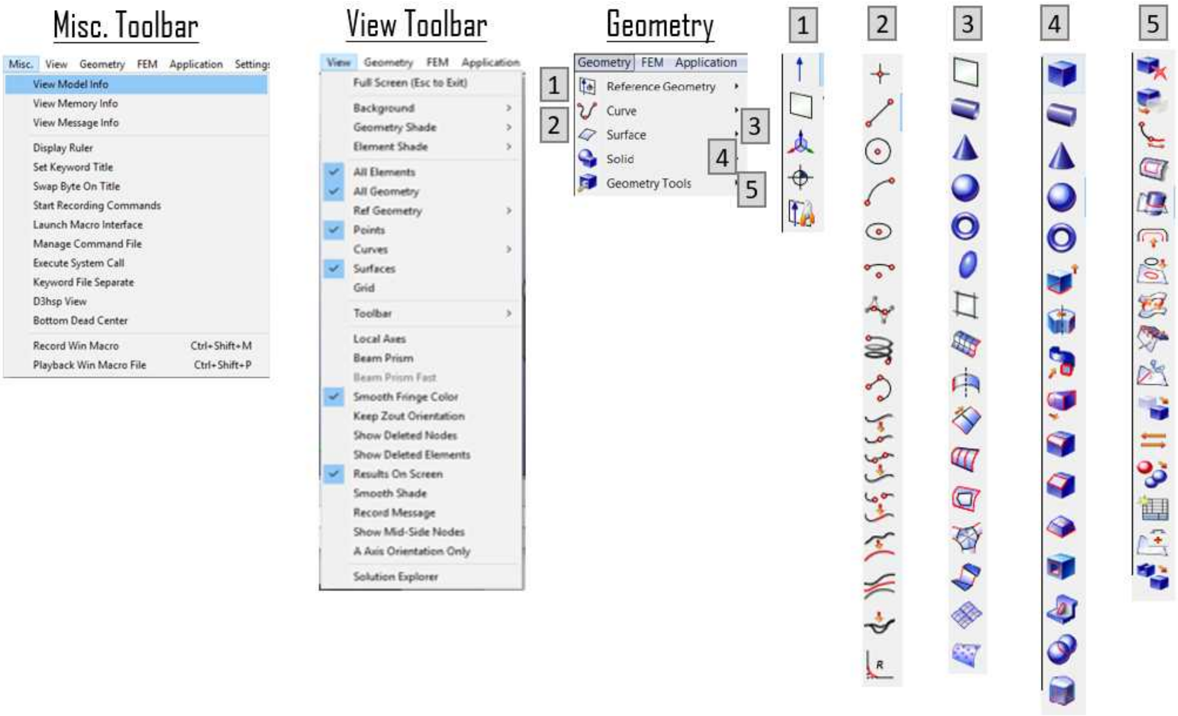
▲ File menu

As you can see in the Import menu, you can import the Dyna keyword model, Nastran model, Ideas model and so on. You can also import a *.stl model of points. Expanding the model tree, you can see that this imported model consists of various surfaces (outer shell).



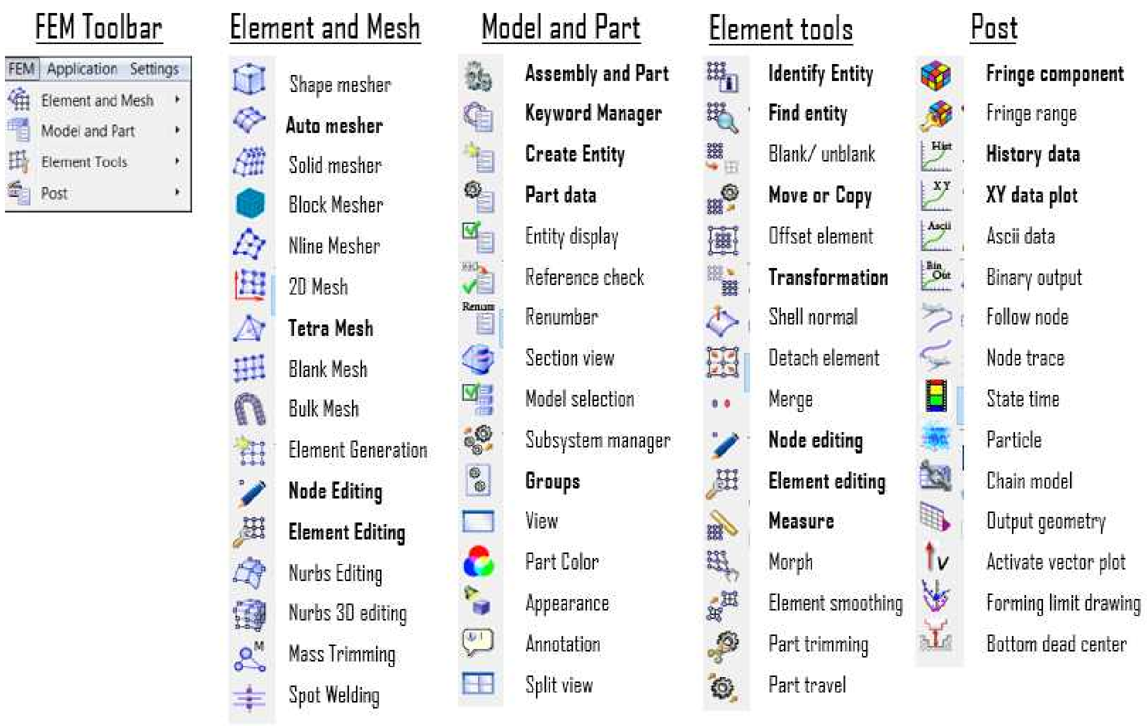
▲ Imported Connecting Rod model

Expand the View menu in the top menu bar to see what updates are done as shown in the graphic display while you use the feature once in the current model. The Geometry toolbar provides basic CAD model functionality with curves, surfaces, solids, and geometry tools. It offers a lot of features, but it takes a lot of time to familiarize and if you already have a CAD system with which you are comfortable, we suggest using it. We find it much more effective to use familiar CAD programs such as CATIA or SolidWorks, which we have already used, rather than geometry modeling in Prepost. In this tutorial, a detailed explanation of geometry modeling is omitted and it is recommended to generate CAD model using a program familiar to the readers.



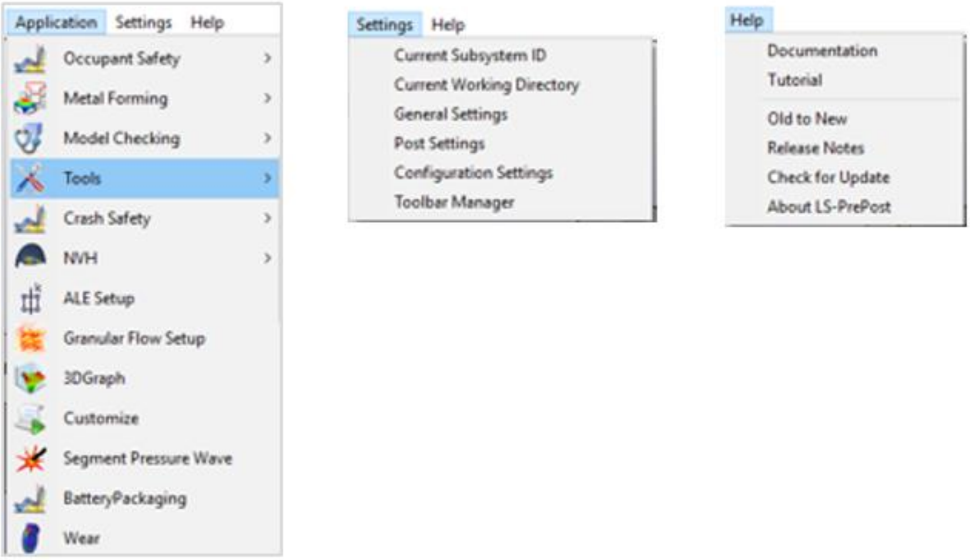
▲ Misc. View. Geometry toolbar

The above figure highlights the most used FEM menu used while modeling. In this FEM menu, you can create, modify and supplement elements and nodes in your own or imported CAD models and create and modify many keywords. The descriptions of each individual menus are shown in the following figure. Commonly and frequently used menus are marked in thick, so remembering the shape and meaning of the icons will increase the efficiency of your work. Specific uses and usages of each option will be covered one by one later, so let's only learn how to use the icons in this chapter.



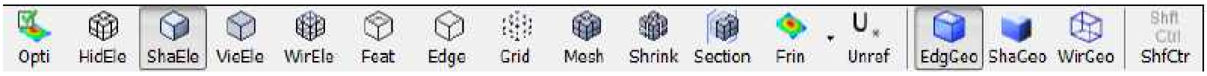
▲ FEM toolbar

In addition, there is **Application**, **Setting** and **Help** menus. The **Application** menu is a GUI function to provide the convenience of specific analysis fields. **Setting** provides a general modeling process setup and options for displaying the post. The **Help** menu provides manuals for modeling with some practical examples.



▲ Menu bar

The following figure is the menu bar present at the bottom of Ls-prepost. The options are mainly related to graphic elements that are used in post-processing during modeling or analysis of results.



Opti Toggle Title, Legend, Min-Max, Time Stamp, Triad, Background Color, Mesh Color, Performance Stats on/off, Feature Tree, ISO View and Animation Play

HidEle Display elements with hidden lines removed

ShaEle Display elements in shaded mode with mesh lines off

VieEle Display elements in plain color mode

WirEle Display elements in wireframe mode

Feat Display elements in feature line mode (default angle=30°)

Edge Display elements in edge line mode

Grid Display each nodal point as a colored pixel

Mesh Toggle element mesh on/off

Shrink Draw elements in shrunken mode (default=0.85)

Section Display section view with plane selected

Frin Toggle Fringe/Line-contours/Iso-surfaces

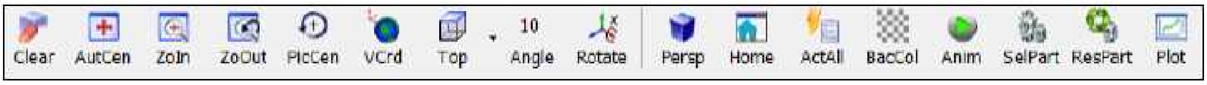
Unref Toggle unreferenced nodes on/off

EdgGeo Display geometry in shaded mode with edges on

ShaGeo Display geometry in shaded mode with edges off

WirGeo Display geometry in wireframe mode

ShfCtr Toggle Off/Shift/Ctrl (for one-handed rotate/pan/zoom)



Clear all picked or highlighted information

AutCen Automatically center model to fit within the window

ZoIn Zoom in, click and drag to draw a box

Zout Zoom out to previous zoom position

Pcen Pick node as a new center point for model rotation

VCrd View coordinate systems

Top Choose Top, Bottom, Front, Back, Right, or Left view

Angle Left-click to reverse rotation direction Right-click to modify rotation angle

Rotate Left-click to rotate about axis shown Right-click to switch rotation axis (X/Y/Z)

Persp Toggle Parallel/Perspective view mode

Home Restore default view and fit model

ActAll Restore all entities to be active

BacCol Toggle background color black/white (Plain background mode only)

Anim Display animation controls or start/stop animation

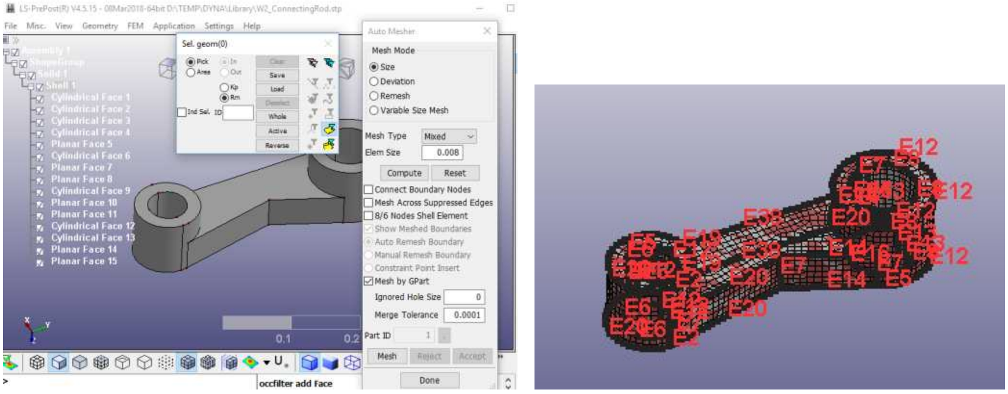
SelPart Launch Assembly and Select Part interface

ResPart Restore the last removed part (Shift+R)

Plot Open XY plot management interface

Now that you have a brief look at the basic features, now let's learn how to use them to create a mesh in a CAD model. Creating a shell element with **Auto Mesher** will create a mesh as shown below.

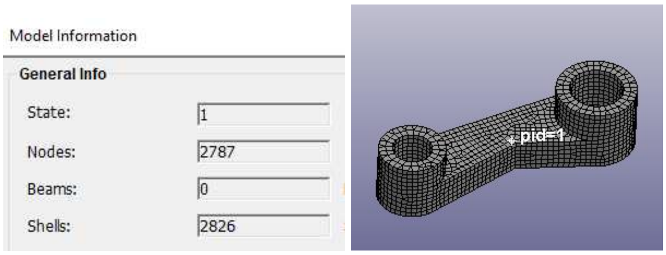
- **Element and mesh → Auto Mesher → Sel. Geom | Pick, select Solid 1 in model tree → Compute → Mesh → Accept**



Mesh Mode: ● Size ○ Deviation ○ Remesh ○ Variable Size Mesh
Mesh Type: Mixed
Elem Size: 0.008
☐ Connect Boundary Nodes
☐ Mesh Across Suppressed Edges
☐ 8/6 Nodes Shell Element
☑ Mesh by GPart
Ignored Hole Size: 0
Merge Tolerance: 0.0001
Part ID: 1

▲ Element generation with Auto mesher

Misc.→ Click on Model Info and you will see that 2787 shell elements have been created. If you uncheck the Geometry part of the model tree, only the Mesh of the model is shown as highlighted below.

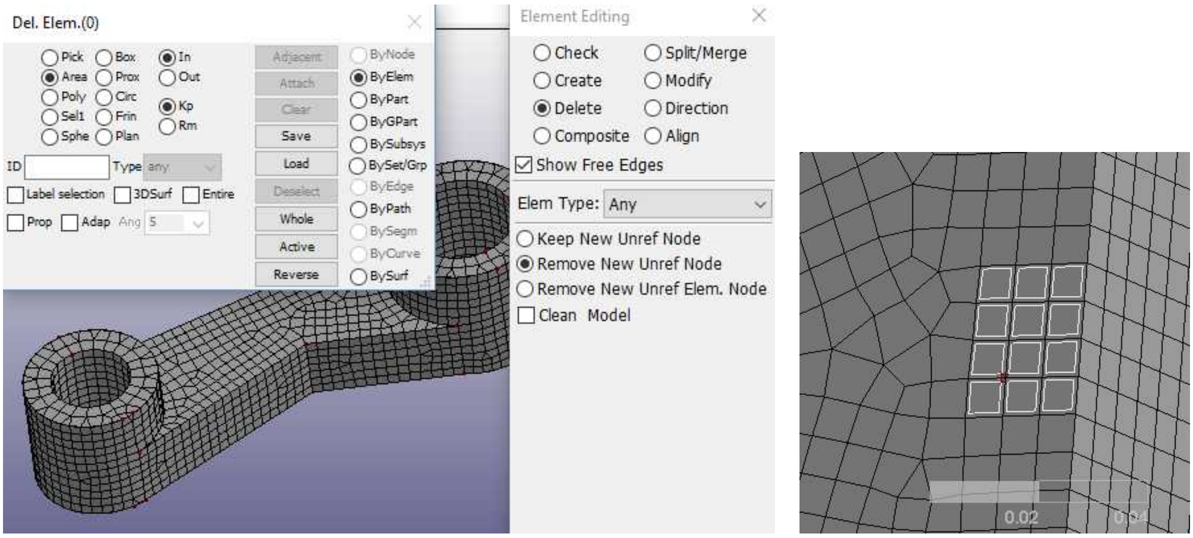

Model Information

General Info

State: 1
Nodes: 2787
Beams: 0
Shells: 2826

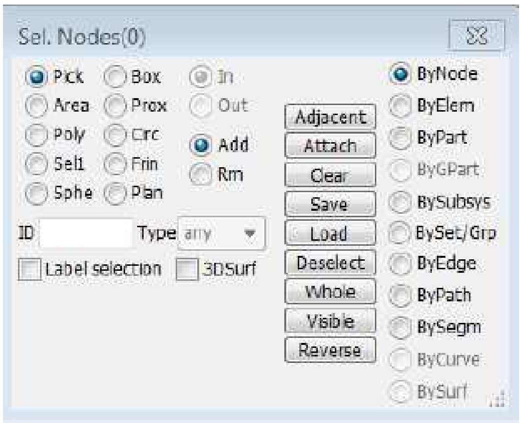
▲ Check model information

Let's delete some of the elements on one side of the rod's connection. Select **Element and Mesh → Element editing → Delete** to display the Selection window. Select **Pick, In, Kp, By Part,** then select the top view at the top of the modeling window and click Delete.



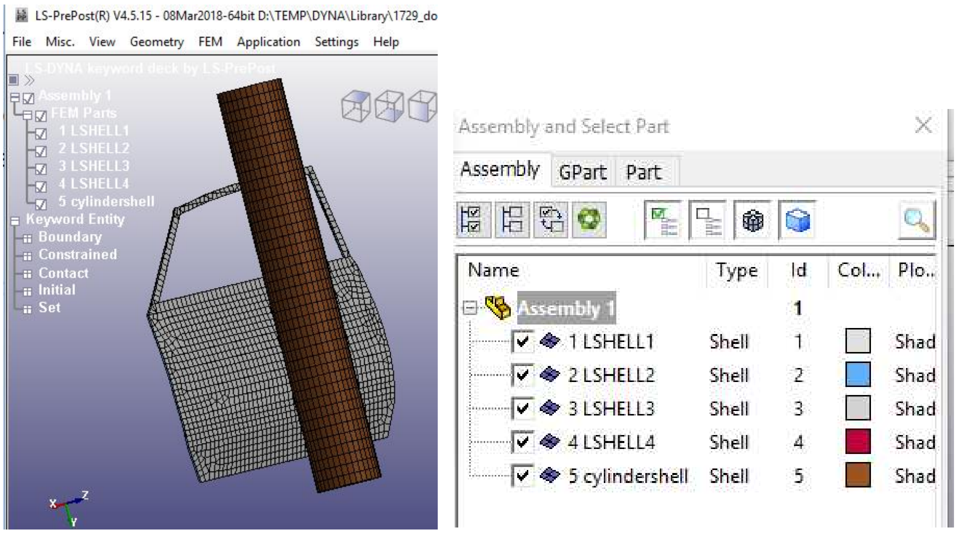
FEM selection function option and element selection

As such, the FEM Selection utility is one of the most frequently used functions during the modeling process, so please refer to the description below to see what functions are available and apply these various functions in the current model.



Adjacent Select adjacent elements
Attach Select attached elements
Clear Clear selection
Save Save selection to buffer or to file
Load Load selection from buffer or from file
Deselect Undo last selection
Whole Select all entities in model
Visible Select all visible entities
Reverse Reverse selection
ByNode Select nodes
ByElem Select elements
ByPart Select parts
BySet Set based selection
ByEdge Edge based selection
ByPath Select first and last along a path
BySegm Select segments
BySurf Select surface (geometry)

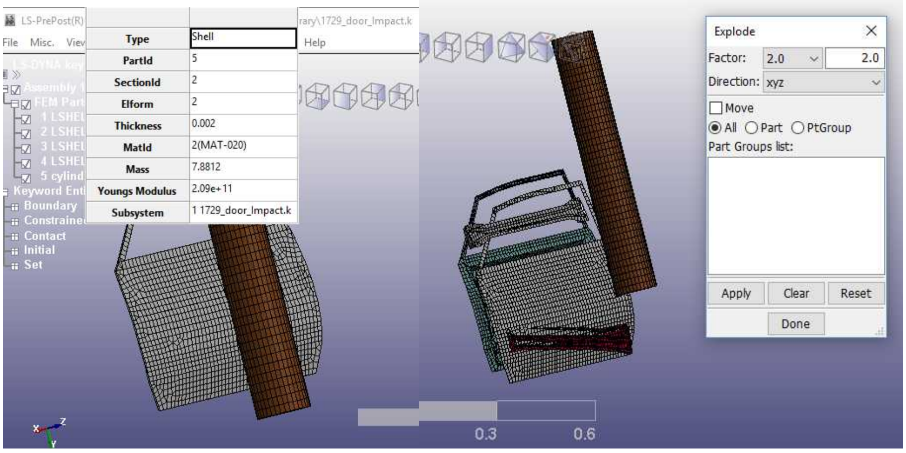
Let's open the ***W2_Door_Impact.k*** model from the attached files work folder. Click **Model and Part** → **Assembly and Select Part** to display a model with five Shell parts. Click Color in the dialog box to change the color of the part and add/delete the view of the model by checking the Activate button.



Name	Type	Id	Col...	Plo..
Assembly 1		1		
☑ 1 LSHELL1	Shell	1	⬜	Shad
☑ 2 LSHELL2	Shell	2	🟦	Shad
☑ 3 LSHELL3	Shell	3	⬜	Shad
☑ 4 LSHELL4	Shell	4	🟥	Shad
☑ 5 cylindershell	Shell	5	🟫	Shad

▲ View options by door impact model parts

If you move the mouse over the part and click the right mouse button, the information of the part is shown as highlighted in the below figure. Click on the **Explode** icon in the lower right corner, select **factor** 2 and then apply and you can easily identify the internal components as the parts are exploded.

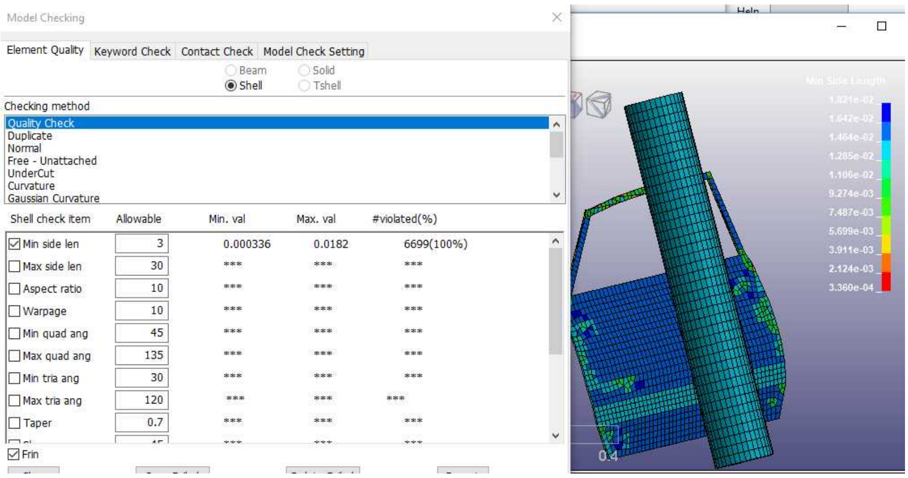


Type	Shell
PartId	5
SectionId	2
Elform	2
Thickness	0.002
MatId	2(MAT-020)
Mass	7.8812
Youngs Modulus	2.09e+11
Subsystem	1 1729_door_Impact.k

Factor: 2.0 2.0
Direction: xyz
☐ Move
● All ○ Part ○ PtGroup
Part Groups list:

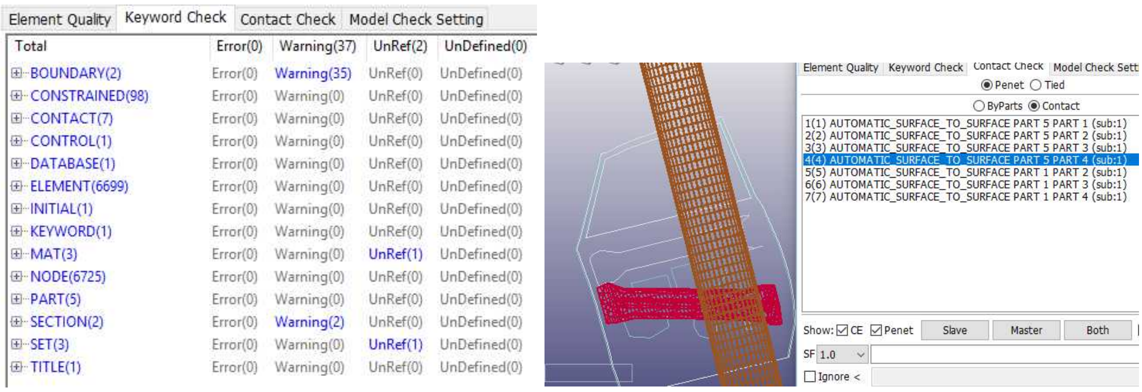
▲ Door Impact model parts information confirmation and Explode implementation screen

Select **Model and Part** → **Keyword Manager** and click on **Model check**. You can check the state of an element with a contour plot by selecting various variables. The figure below shows the distribution of the length of one edge of an element. By selecting **Keyword check** and **Contact check** in the upper tab, it is possible to check whether there is a problem in the model and also it can check whether the contact condition between parts is properly defined. It is very effective for debugging because the **Model check** can review in advance before submitting the calculation for execution. So far, we have looked at the general LS-Prepost function and in the next chapter, we shall create an analysis model using the functions we have seen so far and look at the structure of keywords.



Shell check item	Allowable	Min. val	Max. val	#violated(%)
☑ Min side len	3	0.000336	0.0182	6699(100%)
☐ Max side len	30	***	***	***
☐ Aspect ratio	10	***	***	***
☐ Warpage	10	***	***	***
☐ Min quad ang	45	***	***	***
☐ Max quad ang	135	***	***	***
☐ Min tria ang	30	***	***	***
☐ Max tria ang	120	***	***	***
☐ Taper	0.7	***	***	***

▲ Analyze the state of elements through model check



Total	Error(0)	Warning(37)	UnRef(2)	UnDefined(0)
⊞ BOUNDARY(2)	Error(0)	Warning(35)	UnRef(0)	UnDefined(0)
⊞ CONSTRAINED(98)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ CONTACT(7)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ CONTROL(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ DATABASE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ ELEMENT(6699)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ INITIAL(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ KEYWORD(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ MAT(3)	Error(0)	Warning(0)	UnRef(1)	UnDefined(0)
⊞ NODE(6725)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ PART(5)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ SECTION(2)	Error(0)	Warning(2)	UnRef(0)	UnDefined(0)
⊞ SET(3)	Error(0)	Warning(0)	UnRef(1)	UnDefined(0)
⊞ TITLE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)

```
1(1) AUTOMATIC_SURFACE_TO_SURFACE PART 5 PART 1 (sub:1)
2(2) AUTOMATIC_SURFACE_TO_SURFACE PART 5 PART 2 (sub:1)
3(3) AUTOMATIC_SURFACE_TO_SURFACE PART 5 PART 3 (sub:1)
4(4) AUTOMATIC_SURFACE_TO_SURFACE PART 5 PART 4 (sub:1)
5(5) AUTOMATIC_SURFACE_TO_SURFACE PART 1 PART 2 (sub:1)
6(6) AUTOMATIC_SURFACE_TO_SURFACE PART 1 PART 3 (sub:1)
7(7) AUTOMATIC_SURFACE_TO_SURFACE PART 1 PART 4 (sub:1)
```

▲ Keyword check and contact check

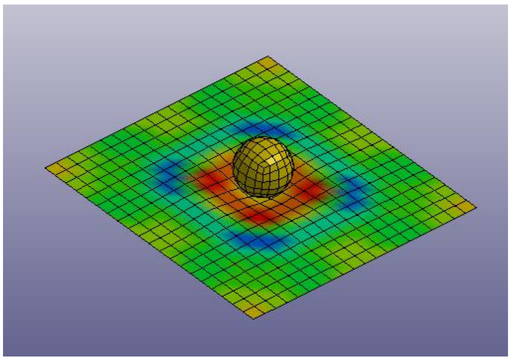
Chapter 3. Simple Impact Analysis: Ball-Plate Drop Analysis

Introduction

If you search on the Internet or any other study materials to learn LS-DYNA for the first time, the most frequently appeared example is the impact analysis of a Ball on the plate. Although it is the most basic interpretation, it is a good example to learn LS-DYNA interpretation procedure or concept. In this chapter, we will practice this simple example step-by-step. If you have previous experience with LS-DYNA, you may skip this chapter.

Reference Model

- W3_SETUP_Ball_Impact.k

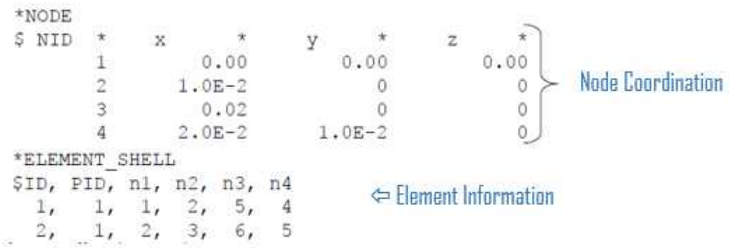


▲ Ball-Plate Drop Impact Analysis Model

The modeling order for LS-DYNA analysis is as follows. The whole process can be done in LS-Prepost and it is recommended that you use a CAD program and the pre-processor that you are familiar with. Typical pre-processors that are compatible with LS-DYNA are Hypermesh, ANSYS, and LS-OASYS. Also, most other programs similarly support LS-DYNA.

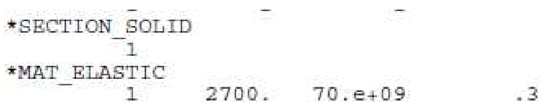
LS-DYNA modeling order

- **Geometry modeling:** Geometry generation for nodes and elements creation, IGES and STP CAD model can be Imported or create geometry models into LS-Prepost.
- **Create Mesh:** Create Node and Element Information



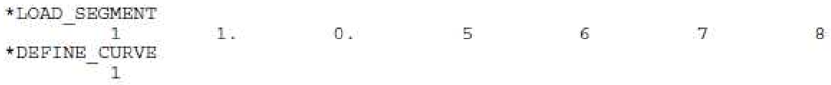
```
*NODE
$ NID  *     x       *       y      *       z      *
       1        0.00           0.00           0.00
       2      1.0E-2              0              0
       3        0.02              0              0
       4      2.0E-2         1.0E-2              0
*ELEMENT_SHELL
$ID, PID, n1, n2, n3, n4
  1,   1,  1,  2,  5,  4
  2,   1,  2,  3,  6,  5
```

- **Set material properties and section properties:** Select property model and select type of section element



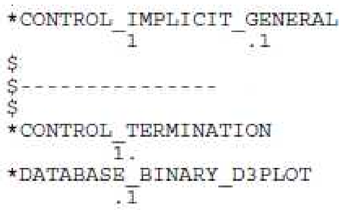
```
*SECTION_SOLID
         1
*MAT_ELASTIC
         1     2700.    70.e+09         .3
```

- **Setting load condition / boundary condition / initial condition**



```
*LOAD_SEGMENT
         1         1.         0.         5         6         7         8
*DEFINE_CURVE
         1
```

- **Set analysis conditions:** set contact conditions, analysis time, result output data, etc.

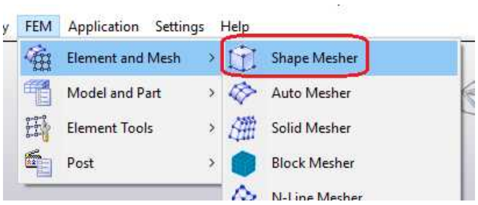


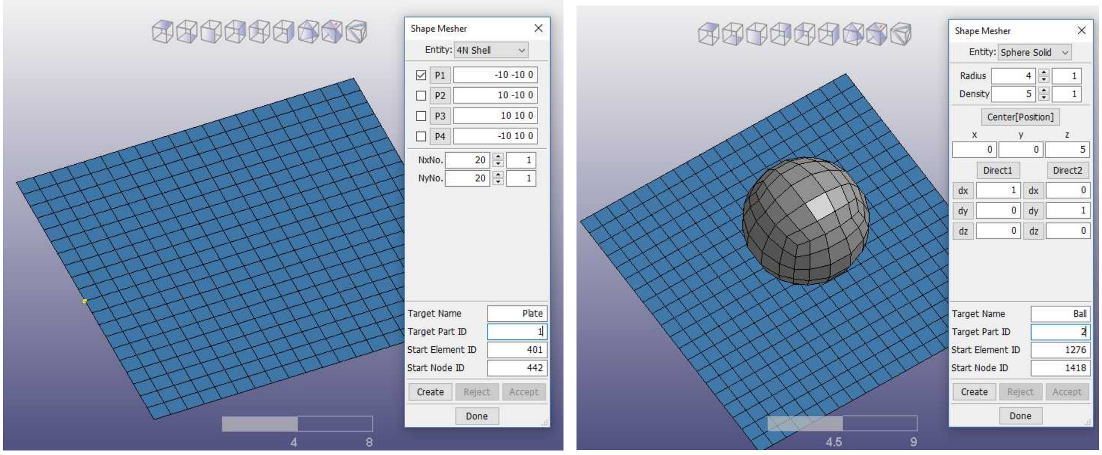
```
*CONTROL_IMPLICIT_GENERAL
         1        .1
$
$---------------
$
*CONTROL_TERMINATION
        1.
*DATABASE_BINARY_D3PLOT
        .1
```

- **Save as * .k (key) file**
- **Generate result file by solving the .k file with LS-DYNA solver**
- **Post-processing**

Step 1 Geometry and Mesh

In LS-Prepost, we will create the mesh of the plate and sphere and then enter the analysis conditions in turn. Open LS-Prepost and select **FEM → Element and Mesh → Shape Mesher** from the top menu bar.




Shape Mesher
Entity: 4N Shell
☑ P1 -10 -10 0
☐ P2 10 -10 0
☐ P3 10 10 0
☐ P4 -10 10 0
NxNo. 20 1
NyNo. 20 1
Target Name Plate
Target Part ID 1
Start Element ID 401
Start Node ID 442

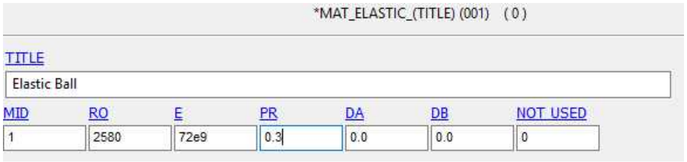
Shape Mesher
Entity: Sphere Solid
Radius 4 1
Density 5 1
Center[Position] x 0 y 0 z 5
Direct1 dx 1 dy 0 dz 0
Direct2 dx 0 dy 1 dz 0
Target Name Ball
Target Part ID 2
Start Element ID 1276
Start Node ID 1418

▲ Ball-Plate Drop Impact Analysis Model

Step 2 Material Properties

The next step is to create a property definition for each part. LS-DYNA provides nearly 300 physical property models. Depending on the material type of the parts to be analyzed, the user can select the choice of material formulation to exactly replicate the physical behavior in the real world.

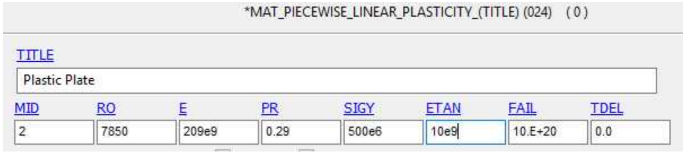
The plate is created with the basic iron-based simple elastomeric property **MAT001_Elastic** and the sphere with the **MAT024_Piecewise_Linear_Plasticity** property model having the aluminum properties. Enter the following property values by selecting the **FEM → Model and Part → Keyword Manager → MAT001 and MAT024** from the top menu.


*MAT_ELASTIC_(TITLE) (001) (0)

TITLE
Elastic Ball

MID	RO	E	PR	DA	DB	NOT USED
1	2580	72e9	0.3	0.0	0.0	0

▲ MAT001_Elastic property input



*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (0)

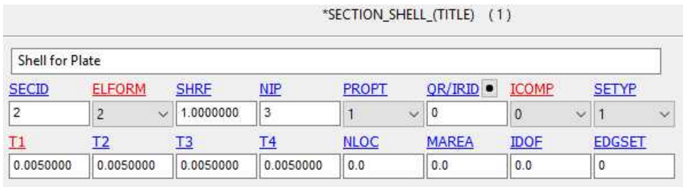
TITLE
Plastic Plate

MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
2	7850	209e9	0.29	500e6	10e9	10.E+20	0.0

▲ MAT024_ Piecewise_Linear_Plasticity Input

Step 3 Section Properties

Section properties are used to describe the physical properties and characteristics of a component such as its thickness, spring stiffness and most importantly the formulation of the elements. Enter the Shell section for the plate and the Solid section for the ball as shown below. Select **Keyword Manager** → **Section** → **Shell** and enter the following values. Likewise, select **Section** → **Solid** to define the property value.

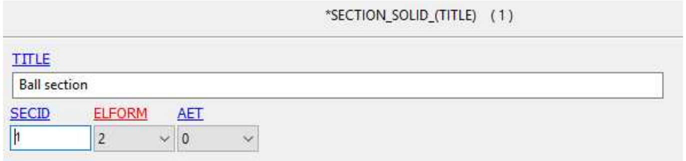


*SECTION_SHELL_(TITLE) (1)

Shell for Plate

SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
2	2	1.0000000	3	1	0	0	1
T1	**T2**	**T3**	**T4**	**NLOC**	**MAREA**	**IDOF**	**EDGSET**
0.0050000	0.0050000	0.0050000	0.0050000	0.0	0.0	0.0	0

▲ Shell Section Input



*SECTION_SOLID_(TITLE) (1)

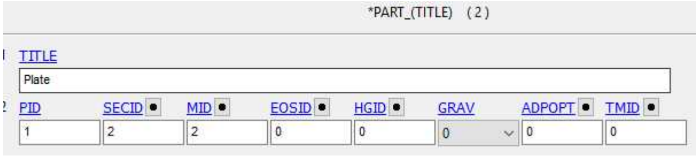
TITLE
Ball section

SECID	ELFORM	AET
1	2	0

▲ Solid Section Input

Step 4 Part Properties

This step is to define the property value and section characteristics for each part. After selecting **Keyword Manager** → **Part** → **Define the property** and section characteristics of Plate and Ball as follows.

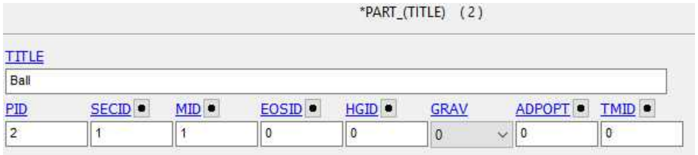


*PART_(TITLE) (2)

TITLE
Plate

PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
1	2	2	0	0	0	0	0

▲ Plate Part Properties



*PART_(TITLE) (2)

TITLE

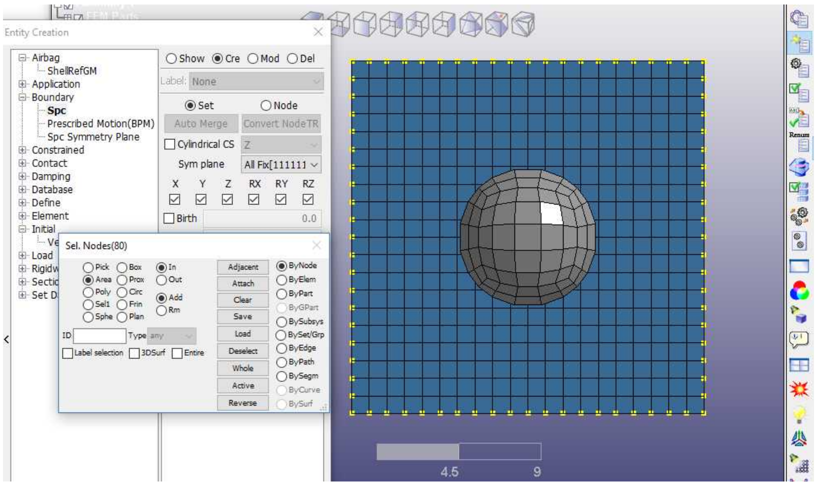
Ball

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
2	1	1	0	0	0	0	0

▲ Ball Part Properties

Step 5 Boundary and Loading Conditions

To set the plate boundary conditions, select **FEM** → **Model and Part** → **Create Entity** → **Boundary** → **SPC**. After selecting all edges of the plate as shown below, check x, y, z, rx, ry, rz and fix all DOF.



Entity Creation

○ Show ● Cre ○ Mod ○ Del

Label: None

● Set ○ Node

☐ Cylindrical CS Z

Sym plane All Fix[111111]

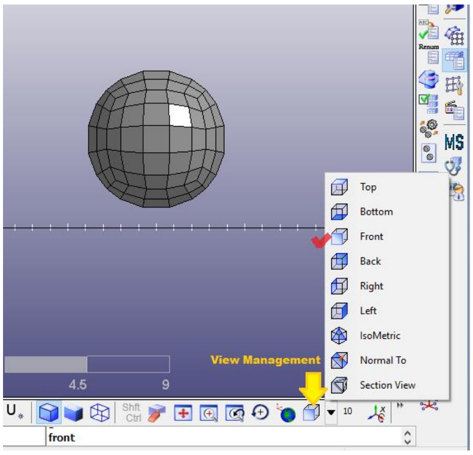
X	Y	Z	RX	RY	RZ
☑	☑	☑	☑	☑	☑

☐ Birth 0.0

Sel. Nodes(80)

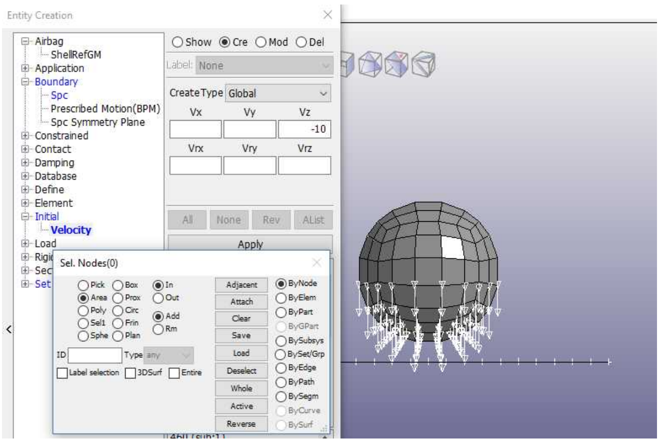
▲ Boundary condition, SPC setup

When aligning the model to select a specific part of the model, you can easily align the model by using the view management function as shown below.



▲ View Screen Setup

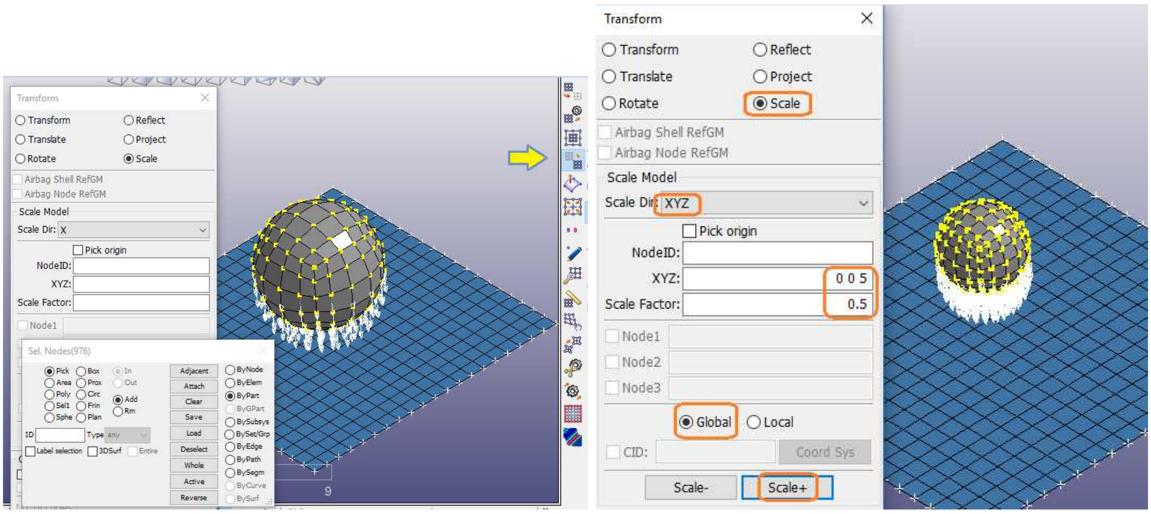
The next step is to define the initial velocity of the sphere. Select **FEM → Model and Part → Create Entity → Initial → Velocity and set Vz = -10m / s** as shown below.



Create Type: Global
Vx: , Vy: , Vz: -10

▲ Initial_Velocity setting

The following is a tip for modifying the model. If the sphere is too large and you want to reduce the diameter to ½, select **FEM → Element tools → Transformation → Scale** and select the whole ball and enter 0.5 for the scale factor as shown below. As shown in the figure below, the diameter of the ball is reduced to half. Transformation is a useful function that can modify various parts such as shrinking, moving and rotating.

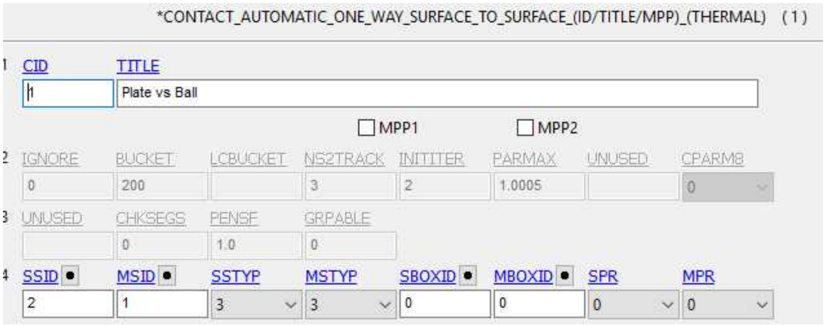


Scale Dir: XYZ
NodeID:
XYZ: 0 0 5
Scale Factor: 0.5

▲ Scale down setting to shrink the ball part

Step 6 Contact Conditions

This step is the setting for the most important contact condition in collision analysis. LS-DYNA provides more than 50 contact condition setting functions. As in this example, a simple collision between ball and plate can be defined using ***Contact_Automatic_ One_Way_Surface_ To_Surface**. Select **Keyword Manager → Contact → Automatic One way Surface to Surface** and set Ball part to Slave and Plate part to Master as shown below.

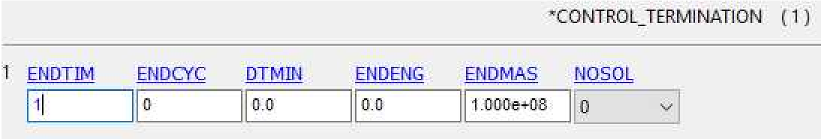


*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (1)

	CID	TITLE						
1	1	Plate vs Ball						
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	2	1	3	3	0	0	0	0

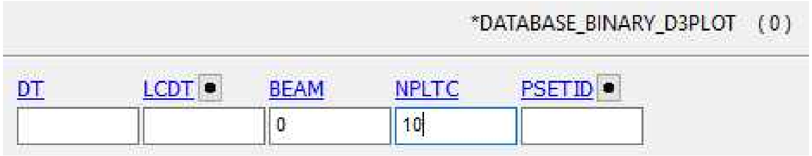
Step 7 Analysis Setup

The final step of modeling is to set the analysis time, various control variables and analysis result variables. Enter **ENDTIM** = 1sec in **Keyword Manager → Control → Termination**. ENTIM is a variable that means the analysis time and it is analyzed for 1 second at 10m / s, so the analysis proceeds as much as the time that the ball moves about 1m. Binary format **D3PLOT** that shows the analysis result is set to save 10 times during the analysis time. Enter NPLTC = 10 in **Keyword Manager → Database → Binary_D3PLOT**.



*CONTROL_TERMINATION (1)

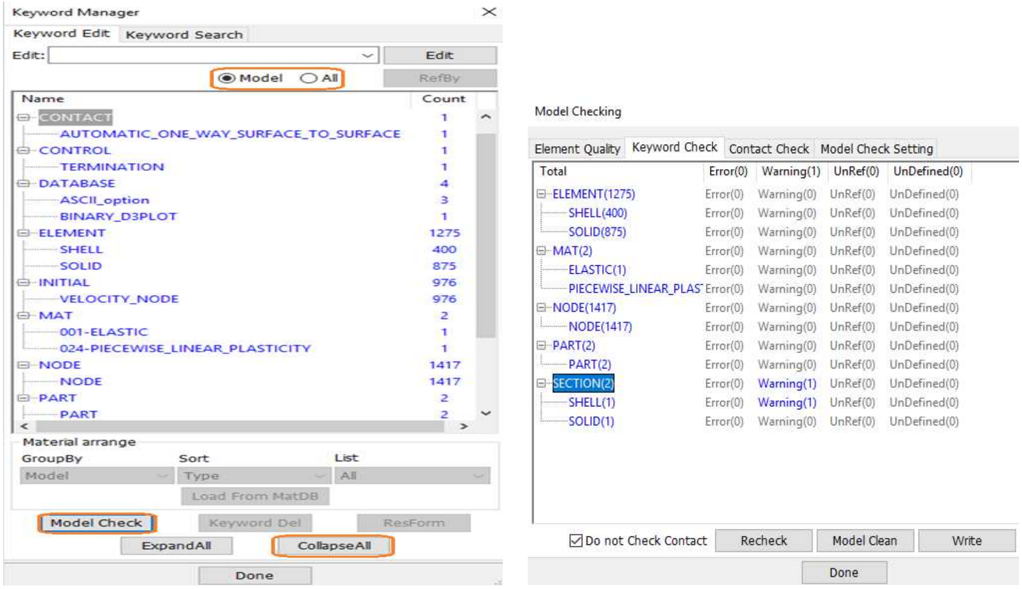
	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS	NOSOL
1	1	0	0.0	0.0	1.000e+08	0



*DATABASE_BINARY_D3PLOT (0)

DT	LCDT ●	BEAM	NPLTC	PSETID ●
		0	10	

In **Keyword Manager → Database → ASCII**, enter **DT** = 0.1 and select **GLSTAT** and **MATSUM** variables. It means the period to save numerical data in ASCII format. When modeling is completed, select Keyword Manager and click **Model Check** as shown below to check and show whether there is an error in overall modeling. Checking once before executing the analysis and checking for errors will effectively manage the working time. When modeling is complete, save the input file as **File → Save As →** ***W3_test.k***.



Name	Count
CONTACT	1
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE	1
CONTROL	1
TERMINATION	1
DATABASE	4
ASCII_option	3
BINARY_D3PLOT	1
ELEMENT	1275
SHELL	400
SOLID	875
INITIAL	976
VELOCITY_NODE	976
MAT	2
001-ELASTIC	1
024-PIECEWISE_LINEAR_PLASTICITY	1
NODE	1417
NODE	1417
PART	2
PART	2

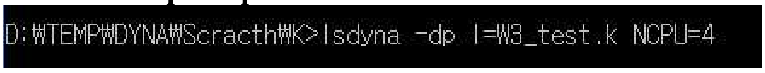
Total	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)
ELEMENT(1275)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
SHELL(400)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
SOLID(875)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
MAT(2)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
ELASTIC(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
PIECEWISE_LINEAR_PLAS	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
NODE(1417)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
NODE(1417)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
PART(2)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
PART(2)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
SECTION(2)	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)
SHELL(1)	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)
SOLID(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)

▲ Model Check in Keyword Manager

Step 8 Job Execution

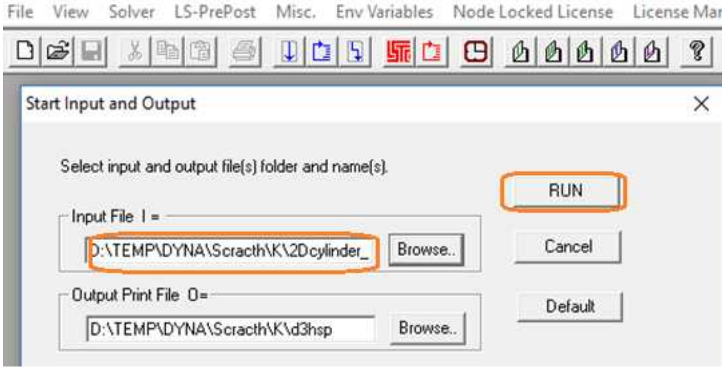
Once the modeling is completed, the analysis step is executed. Depending on the readers' environment, the analysis can be performed in various ways. It can be executed by the use of command in the command window, or it can be executed in LS-DYNA Manager and also the task can be executed and managed directly in LS-Prepost.

- **Command prompt**



```
D:₩TEMP₩DYNA₩Scracth₩K>lsdyna -dp I=W3_test.k NCPU=4
```

- **LS-DYNA Manager**



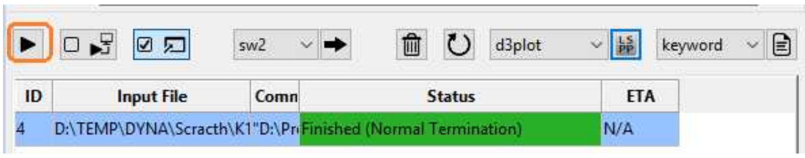
Start Input and Output

Select input and output file(s) folder and name(s).

Input File I = D:\TEMP\DYNA\Scracth\K\2Dcylinder_

Output Print File O= D:\TEMP\DYNA\Scracth\K\d3hsp

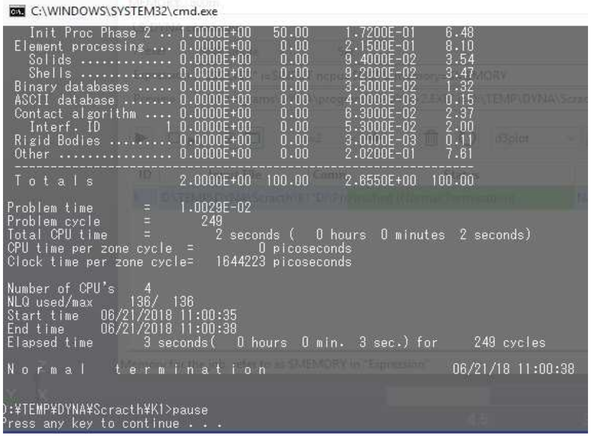
- **LS-Prepost → File → LS-Run**



ID	Input File	Comn	Status	ETA
4	D:\TEMP\DYNA\Scracth\K1"D:\Pr		Finished (Normal Termination)	N/A

Step 9 Result Visualization

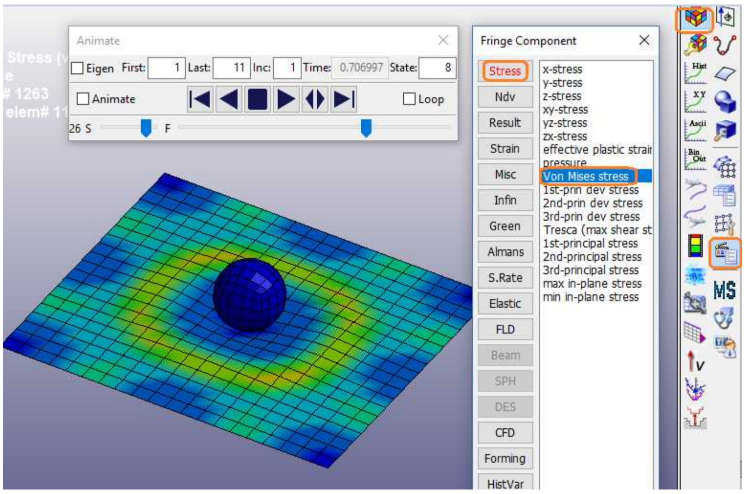
As the analysis proceeds, the log screen as shown in the figure below appears and when Normal termination appears, it means that the process is completed normally. This log screen records the overall contents of the analysis process, so it can be useful for future debugging.



```
   Init Proc Phase 2 .. 1.0000E+00   50.00     1.7200E-01    6.48
 Element processing ... 0.0000E+00    0.00     2.1500E-01    8.10
   Solids ............. 0.0000E+00    0.00     9.4000E-02    3.54
   Shells ............. 0.0000E+00    0.00     9.2000E-02    3.47
 Binary databases ..... 0.0000E+00    0.00     3.5000E-02    1.32
 ASCII database ....... 0.0000E+00    0.00     4.0000E-03    0.15
 Contact algorithm .... 0.0000E+00    0.00     6.3000E-02    2.37
   Interf. ID         1 0.0000E+00    0.00     5.3000E-02    2.00
 Rigid Bodies ......... 0.0000E+00    0.00     3.0000E-03    0.11
 Other ................ 0.0000E+00    0.00     2.0200E-01    7.61
 ----------------------------------------------------------------
 T o t a l s            2.0000E+00  100.00     2.6550E+00  100.00

Problem time       =    1.0029E-02
Problem cycle      =       249
Total CPU time     =         2 seconds (   0 hours  0 minutes  2 seconds)
CPU time per zone cycle  =         0 picoseconds
Clock time per zone cycle=   1644223 picoseconds

Number of CPU's    4
NLQ used/max      136/  136
Start time    06/21/2018 11:00:35
End time      06/21/2018 11:00:38
Elapsed time       3 seconds(   0 hours  0 min.  3 sec.) for     249 cycles

N o r m a l    t e r m i n a t i o n                      06/21/18 11:00:38

D:¥TEMP¥DYNA¥Scracth¥K1>pause
Press any key to continue . . .
```

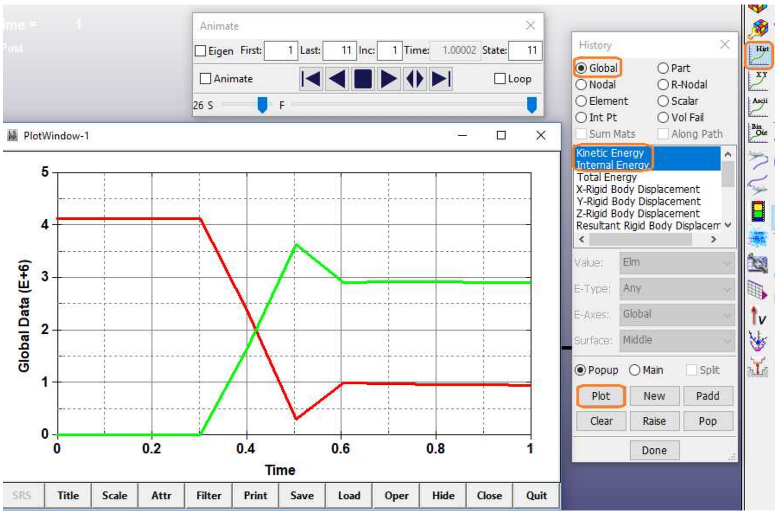
▲ Log Screen

In Prepost, choose **File** → **Open** → **Binary Plot** and select the **D3PLOT** file in the resulting folder. If you select **Fringe component** and click Von Mises stress in Stress as shown below, the analysis result of stress distribution is displayed. In the **Animate bar**, adjust the buttons to see the results of the 10 frames. In addition, let's click on various analysis result variables once to see what results are displayed.



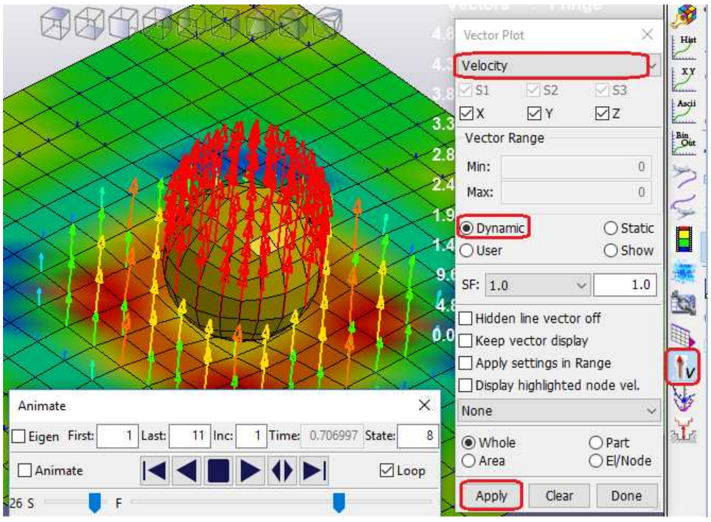
▲ Von Mises Stress Distribution Results

The next step is to check the **History output**. Let's check the energy distribution by selecting History output and plotting Kinetic Energy and Internal Energy of the entire model as shown below. The energy distribution graph can be saved as numerical data by clicking **Save** and used in reports or papers. In addition, various useful functions for generating reports are provided.



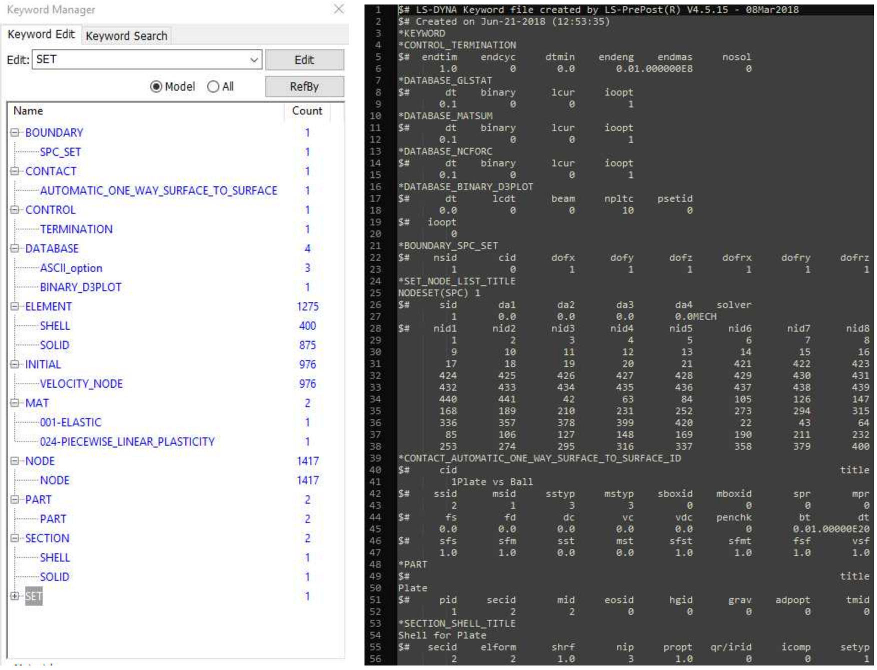
▲ History Energy distribution graph

Another useful feature is the ability to display the result of the analysis as a vector, let's demonstrate the velocity vector of the ball as shown below. It is a feature that can greatly improve the visibility of ball movement and it is a very useful function to observe the tendency of fluid flow in fluid analysis.



▲ Velocity Vector plot

So far, we have practiced a simple example of a collision between ball and plate. If this example was the first LS-DYNA modeling, I hope that this exercise will give you an idea of the big flow of overall modeling and result interpretation. Now let's go back to Prepost and expand the Model Keyword in the **Keyword Manager** to see what keywords we have created. Then open the * .k file in the modeling folder with a text editor to compare each other.



Name	Count
BOUNDARY	1
SPC_SET	1
CONTACT	1
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE	1
CONTROL	1
TERMINATION	1
DATABASE	4
ASCII_option	3
BINARY_D3PLOT	1
ELEMENT	1275
SHELL	400
SOLID	875
INITIAL	976
VELOCITY_NODE	976
MAT	2
001-ELASTIC	1
024-PIECEWISE_LINEAR_PLASTICITY	1
NODE	1417
NODE	1417
PART	2
PART	2
SECTION	2
SHELL	1
SOLID	1
SET	1

```
$# LS-DYNA Keyword file created by LS-PrePost(R) V4.5.15 - 08Mar2018
$# Created on Jun-21-2018 (12:53:35)
*KEYWORD
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas     nosol
       1.0         0       0.0       0.01.000000E8         0
*DATABASE_GLSTAT
$#      dt    binary      lcur     ioopt
       0.1         0         0         1
*DATABASE_MATSUM
$#      dt    binary      lcur     ioopt
       0.1         0         0         1
*DATABASE_NCFORC
$#      dt    binary      lcur     ioopt
       0.1         0         0         1
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
       0.0         0         0        10         0
$#   ioopt
         0
*BOUNDARY_SPC_SET
$#    nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1         1
*SET_NODE_LIST_TITLE
NODESET(SPC) 1
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0MECH
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8
         9        10        11        12        13        14        15        16
        17        18        19        20        21       421       422       423
       424       425       426       427       428       429       430       431
       432       433       434       435       436       437       438       439
       440       441        42        63        84       105       126       147
       168       189       210       231       252       273       294       315
       336       357       378       399       420        22        43        64
        85       106       127       148       169       190       211       232
       253       274       295       316       337       358       379       400
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_ID
$#     cid                                                                 title
         1Plate vs Ball
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         2         1         3         3         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0       0.0         0       0.01.00000E20
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
       1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
*PART
$#                                                                         title
Plate
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         2         2         0         0         0         0         0
*SECTION_SHELL_TITLE
Shell for Plate
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2         2       1.0         3       1.0         0         0         1
```

▲ Keyword Manager vs. Model.k

Since there are often opportunities to review keywords directly in the modeling process in the future, it is helpful to learn the concepts of the structure of input keywords and the relationship between keywords generated in LS-Prepost. We will look more closely at the keyword structure in the next chapter.

KEYWORD STRUCTURE

```
$# LS-DYNA Keyword file created by LS-PrePost(R) V4.5.15 - 08Mar2018
$# Created on Jun-21-2018 (12:53:35)
*KEYWORD
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas     nosol
       1.0         0       0.0       0.01.000000E8         0
*DATABASE_GLSTAT
$#      dt    binary      lcur     ioopt
       0.1         0         0         1
*DATABASE_MATSUM
$#      dt    binary      lcur     ioopt
       0.1         0         0         1
*DATABASE_NCFORC
$#      dt    binary      lcur     ioopt
       0.1         0         0         1
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
       0.0         0         0        10         0
$#   ioopt
         0
*BOUNDARY_SPC_SET
$#    nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         1         1         1         1         1         1
*SET_NODE_LIST_TITLE
NODESET(SPC) 1
$#     sid       da1       da2       da3       da4    solver
         1       0.0       0.0       0.0       0.0MECH
$#    nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
         1         2         3         4         5         6         7         8
 --------
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_ID
$#     cid                                                                 title
         1Plate vs Ball
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         2         1         3         3         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0       0.0         0       0.01.00000E20
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
       1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
*PART
$#                                                                         title
Plate
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         2         2         0         0         0         0         0
*SECTION_SHELL_TITLE
Shell for Plate
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         2         2       1.0         3       1.0         0         0         1
$#      t1        t2        t3        t4      nloc     marea      idof    edgset
     0.005     0.005     0.005     0.005       0.0       0.0       0.0         0
*MAT_PIECEWISE_LINEAR_PLASTICITY_TITLE
Plastic Plate
$#     mid        ro         e        pr      sigy      etan      fail      tdel
         2    7850.02.09000E11       0.295.000000E81.00000E101.00000E21       0.0
$#       c         p      lcss      lcsr        vp
```

Model Keyword Title and Basic Information

Analysis end time ENDTIM

ASCII analysis result request

Request results in Binary

Setup BC

Sectup Contact

Define Part Characteristic

Define Section Characteristic

Material Properties

```
     0.0     0.0       0       0     0.0
$#   eps1    eps2    eps3    eps4    eps5    eps6    eps7    eps8
     0.0     0.0     0.0     0.0     0.0     0.0     0.0     0.0
$#    es1     es2     es3     es4     es5     es6     es7     es8
     0.0     0.0     0.0     0.0     0.0     0.0     0.0     0.0
*PART
$#                                                         title
Ball
$#    pid   secid     mid   eosid    hgid    grav  adpopt    tmid
        2       1       1       0       0       0       0       0
*SECTION_SOLID_TITLE
Ball section
$#  secid  elform     aet
       1       2       0
```

Setup Section(Solid)

```
*MAT_ELASTIC_TITLE
Elastic Ball
$#    mid      ro       e      pr      da      db  not used
       1   2580.07.20000E10     0.3     0.0     0.0       0
```

Material properties

```
*INITIAL_VELOCITY_NODE
$#    nid      vx      vy      vz     vxr     vyr     vzr    icid
      442     0.0     0.0   -10.0     0.0     0.0     0.0       0
 ----------------------
```

Setup initial velocityh

```
*ELEMENT_SOLID
$#   eid    pid    n1    n2    n3    n4    n5    n6    n7    n8
     401      2   442   478   484   448   443   479   485   449
 ----------------------
```

Solid element info.

```
*ELEMENT_SHELL
$#   eid    pid    n1    n2    n3    n4    n5    n6    n7    n8
       1      1    22    23     2     1     0     0     0     0
----------------------
```

Shell element info.

```
*NODE
$#   nid               x               y               z      tc      rc
       1           -10.0           -10.0             0.0       0       0
-----------------------
*END
```

Node info.

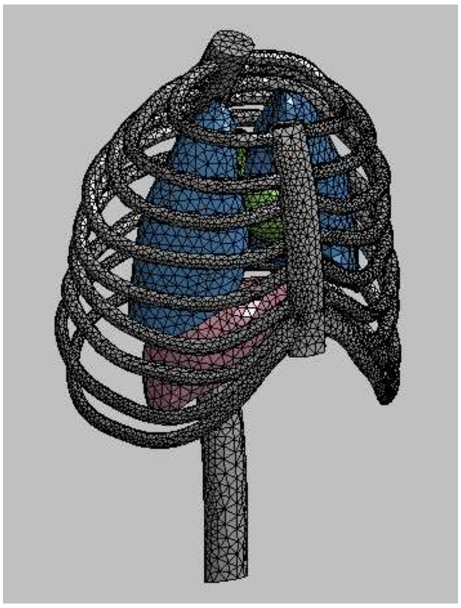
If you look at the keywords and connect them to what you've done in Prespost, you'll understand the concept of how LS-DYNA's keywords are generated in Prepost.

Chapter 4. Element and Mesh Generation

Elements in LS-DYNA

LS-Dyna provides a wide diversity of element formulation from which the user can select accordingly. Various kinds of mesh free elements, particle elements and cohesive elements, which are used for large deformation analysis, including basic beams, shells and solid elements are also provided.

Engineers or graduate students who use LS-DYNA and having some experience with FEM, so the detailed explanation of basic elements and nodes are skipped. The following figure is the human torso modeling the author created for the ballistic analysis of bulletproof projects in graduate school. Various elements are applied to meet the characteristics of the organs like lungs, heart, and kidneys.



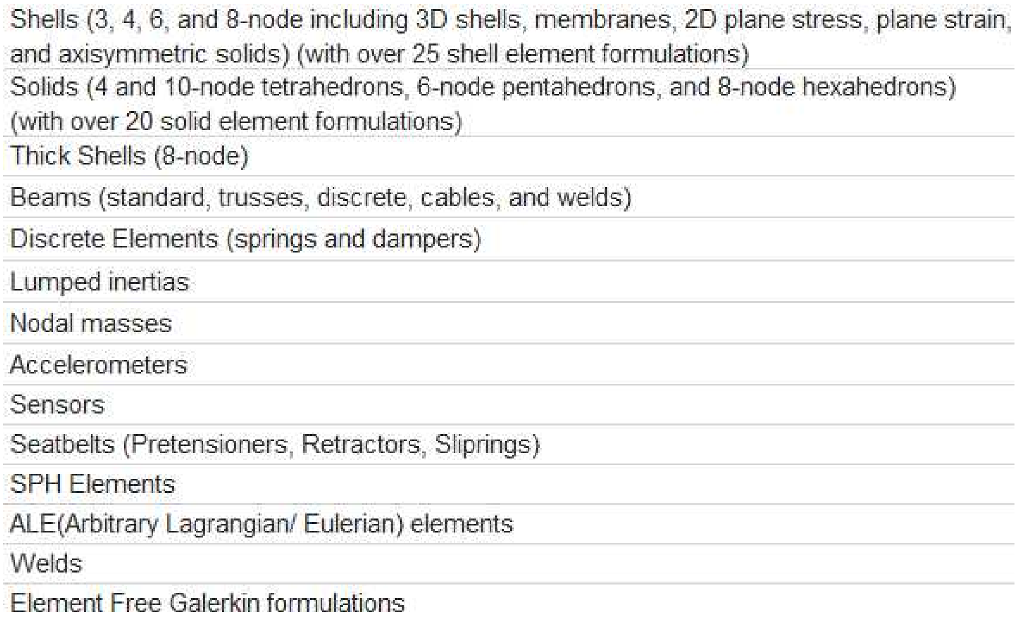
▲ Elements in Human torso model[5]

---

[5] Image courtesy of BW engineering, 2018

LS-DYNA Element library

LS-DYNA is a general-purpose analysis program that can use many elements. It is up to the analyst's experience and know-how to make the right choice of exactly which element to use from among so many element types available. There are many people who think that a very small mesh in 3D element modeling will produce good results without any worries. However, in fact, experienced engineers know that the shell model is much more reliable and effective than solid elements in vehicles and aircraft structures. The element library provided by LS-DYNA is as follows.



Shells (3, 4, 6, and 8-node including 3D shells, membranes, 2D plane stress, plane strain, and axisymmetric solids) (with over 25 shell element formulations)
Solids (4 and 10-node tetrahedrons, 6-node pentahedrons, and 8-node hexahedrons) (with over 20 solid element formulations)
Thick Shells (8-node)
Beams (standard, trusses, discrete, cables, and welds)
Discrete Elements (springs and dampers)
Lumped inertias
Nodal masses
Accelerometers
Sensors
Seatbelts (Pretensioners, Retractors, Sliprings)
SPH Elements
ALE(Arbitrary Lagrangian/ Eulerian) elements
Welds
Element Free Galerkin formulations

Essentially, there are four criteria for choosing the right selection of elements in FEM analysis along with the experience of the engineer.

- **What shape model (2D? 3D?) Will you use to analysis?**
- **How accurate do you need to analysis?**
- **How quickly should the analysis be completed?**
- **What properties will be applied?**

The rule of thumb that can be applied when performing an analysis task on a specific model is the "***Crawl-Walk-Run***" Rule. It means to look at the geometry to be analyzed first, apply the simplest form of an element that can be thought of and apply the advanced element as you build the model gradually.

For example, if a structure having more than 10-meter width and length is a truss structure consisting of steel rods, the experienced analyst will start modeling using 1D elements (Beam, Truss). If a structure is a cylinder shape and the load and boundary conditions are the same throughout the circumferential direction, the 2D Axisymmetric element will be applied.

Occasionally, choosing between the plane stress element and the plane strain element may be confusing. This is simply the distinction between the characteristics of idealizing the stress distribution of 3D in 2D. Plane stress means that the thickness is very small (t / L << 1) compared to the length and the plane strain is very thick (t ≈ Infinite).

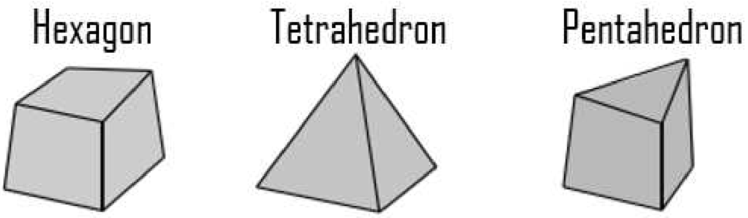
Recent advances in CAD programs will allow Analysis engineers to take over complex 3D models from CAD engineers and then use the auto mesh feature to mesh directly to create FE models comprising very small tetrahedron elements. However, most of these models do not provide effective analysis results. Complex geometry that does not affect any physical behavior shall be simplified. In the case of a thin-walled structure, the shell element must be created by defining the mid-plane. Of course, with the latest FEM preprocessors, including LS-Prepost, often include Midsurface functionality, making them simple to work with. Areas of poor quality of the element always must go through various ways of improving the quality. Next, the element formulation must be selected to be appropriate for the physics to be analyzed. Academic understanding is important, but above all, experience through practice is most important.

Solid Elements

Let's take a quick look at the solid and shell elements that are most commonly dealt with to be realistic within the real world. Solid elements are used to create mesh directly on the target structure without further processing, including simplification and optimization, but it takes a lot of time to generate mesh, computational CPU time and post-processing. The result image may be preferred because it is similar to Realistic and the plot looks good, but from the FEM expert point of view, it is a kind of lazy model. You don't have to worry too much about the modeling process, but it takes a considerable amount of time to follow. However, for a bulky model, solid elements must be used. On the other hand, in thin-walled structures such as the skin structure of an aircraft and the chassis of a vehicle, relatively good results are not obtained. The use of solid elements is typical in high volume structures such as engine blocks, tires, and thick metal tools.

▲ Mechanical parts that require the use of solid elements

Let's take a look at what kind of solid elements are provided in DYNA. The element must be determined in consideration of various factors such as whether the shape of the element like Hexagon, Tetra, or a strong steel series or a soft material such as rubber. Depending on what element you select, the calculation time will be greatly affected.



▲ Element shapes in various forms

The chart below shows the types of solid elements. There are more than 100 types, so you don't have to understand everything, but it's good to be aware of some of the most commonly used elements formulation. Basically, in modeling with solid elements without special concerned analysis, using **Elform** # 1 or # 2 would not be a problem. The details will be explained in an elaborate manner in the exercise where the element is used.

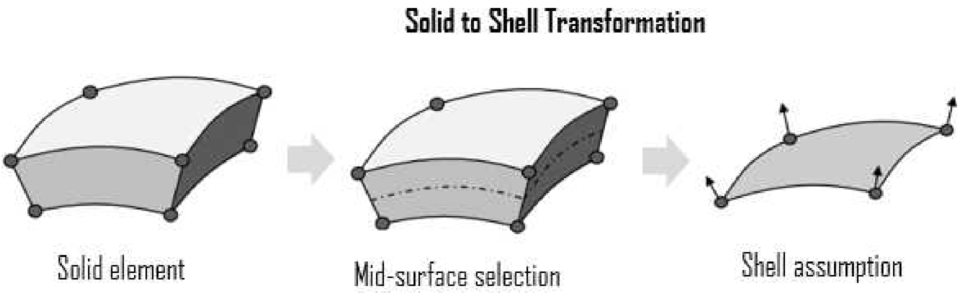
Elform	Description
EQ.-2✓	Fully integrated S/R solid intended for elements with poor aspect ratio, efficient formulation.
EQ.-1	1 point rotational for *MAT_MODIFIED_HONEYCOMB See remark 3.
EQ.0	Constant stress solid element (default),
EQ.2✓	Fully integrated S/R solid. See remark 4 below,
EQ.3✓	Fully integrated quadratic 8 node element with nodal rotations
EQ.4✓	S/R quadratic tetrahedron element with nodal rotations,
EQ.5	1 point ALE,
EQ.6	1 point Eulerian,
EQ.7	1 point Eulerian ambient,
EQ.8	Acoustic,
EQ.9	1 point rotational for *MAT_MODIFIED_HONEYCOMB.
EQ.10✓	1 point tetrahedron.
EQ.11	EQ.11: 1 point ALE multi material element
EQ.12	1 point integration with single material and void.
EQ.13✓	1 point nodal pressure tetrahedron. See remark 14 below.
EQ.14	8 point acoustic
EQ.15	2 point pentahedron element.
EQ.16✓	4 or 5 points 10#noded tetrahedron (See Remark 13)
EQ.17	10 noded composite tetrahedron
EQ.18	8 point enhanced strain solid element for linear statics only
EQ.19	4 point cohesive element
EQ.20	4 point cohesive elements with offsets for use with shells
EQ.115	1 point pentahedron element with hourglass control

▲ Type of solid element of LS-DYNA

If you're dealing with a more complex model, you'll most likely use the Tetrahedron element rather than the Hexagon, in which case **Elform** = 4, 10 or 13 will be fine. If more sophisticated interpretation is required, better results can be obtained by using Elform 16 or 17, the higher-order tetrahedron element.

Shell Elements

The use of shell elements in the finite element analysis is quite high. This is because many machine components are manufactured using thin panels or sheet metals. In particular, the proper application of shell elements is very important in aviation and shipbuilding. Much work and knowledge are required to switch for selecting the appropriate Shell element. In the aerospace sector, it has been established to some extent, while other sectors are known to vary greatly depending on the analyst and the department responsible for the analysis. There are many ways and theories for the application of the Shell element. If there is no special requirement in the company or in a graduate project, it is safe to assume half the thickness as mid-plane. However, if an accurate analysis of the exact bending load is needed, it is important to apply the shell element with the proper theory.



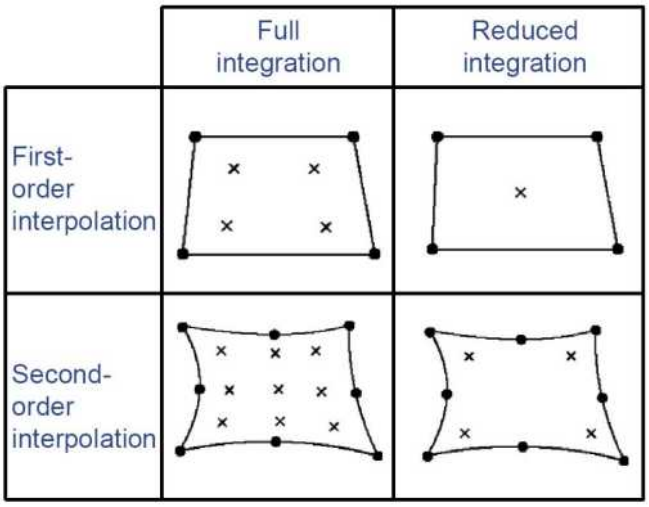
▲ Concept of Switching to the Shell Element

Looking at the DYNA, ABAQUS, or Ansys manuals on shell elements, you can easily identify Kirchhoff-love models, Reissner-Mindlin models and high-order shell theory which are applied for 2D shell formulation. Or, if you have people around you who have experience with FEM, they may be referring to these models. Once you understand it correctly, you won't have any difficulty to apply these based on the requirement. Let's open the solid mechanics textbook studied during the College of Engineering. Kirchhoff and Mindlin plate theory is well documented, so let's review those sectors before getting into the analysis. The following table summarizes the shell elements used in LS-DYNA. In the case of a general shell element, using Elform 2 or 16 should be fine. For models with Hughes-Liu shell, Elfrom 1 and 6 are identified for and for models with Belytschko-Lin-Tsay shell, Elform No. 2 and Elform No. 16 can be applied to a wide range of applications since the general fully integrated shell is applied.

Elform	Description
EQ.1✓	Hughes-Liu with cross section integration (default),
EQ.2✓	Belytschko-Tsay resultant beam (resultant),
EQ.3	truss (resultant), see remark
EQ.4	Belytschko-Schwer full cross#section integration
EQ.5	Belytschko-Schwer tubular beam with cross-section integration
EQ.6✓	S/R Hughes-Liue
EQ.7✓	2D plane strain shell element (xy plane)
EQ.8✓	2D axisymmetric volume-weighted shell element (xy plane)
EQ.9	spotweld beam, see *MAT_SPOTWELD
EQ.11	integrated warped beam
EQ.12	resultant warped beam
EQ.13	small displacement, linear Timoshenko beam with exact stiffness
EQ.14	Axisymmetric solid - area weighted,
EQ.15	Axisymmetric solid - volume weighted,
EQ.16✓	Fully integrated shell element
EQ.17	Fully integrated DKT, triangular shell element
EQ.25✓	Belytschko-Tsay shell with thickness stretch
EQ.26	Fully integrated shell with thickness stretch
EQ.27	triangular shell with thickness stretch

▲ Type of shell element of LS-DYNA

If you look at the element options, the first thing is to set the Gauss Point number. It is important to understand that stress and strain are calculated at the Gauss point which is also called as integration point and the velocity, acceleration, etc. associated with displacement are calculated at nodes. This is a necessary part of understanding the hourglass control later. Most LS-DYNA elements are provided with elements that are under-integrated or reduced integrated. This is most likely the case where there is no full integration in the element formulation description. In short, if there are one or fewer Gaussian points that calculate stress and strain in the element then, the calculation time is faster but the accuracy of the analysis value may be reduced. Theoretically, the larger the number of integral points, the more accurate the data will be.



▲ **Figure 4.5** Reduced Integration vs. Full Integration element

FEM is basically Displacement based method, which calculates stress through displacement. The stiffness matrix of the system is always calculated stiffer than actual behavior. The calculation is somewhat less stiff for a small number of Reduced Integration elements, so in some cases, it is recommended to use **Reduced Integration elements** for better results. This is the case, for example, in the forming analysis where plastic deformation occurs or in the collision analysis where large deformation occurs. This is why many reduced elements are provided in LS-DYNA. Using the reduced integration element has the advantage of shortening the analysis time, but with fewer integration points, in some cases an hourglass mode (or zero-energy mode) occurs where the stiffness matrix becomes zero, resulting in deformations such as hourglasses. In this case, it should be corrected using the Hourglass control Keyword or solved using the Full integration element.

W4

Workshop 4.
Create Element in Geometry Part

In this workshop, we will practice creating and modifying meshes in Geometry models.

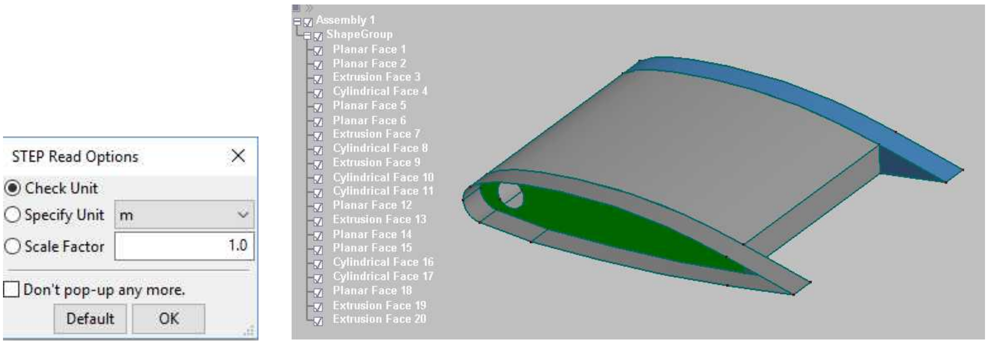
Required model

- W4_WingBox.stp: Wing geometry model
- W4_WingBox_MeshGen.k: Final state model

Creating Elements

Let's import the CAD model entitled *'W5_WingBox.stp'* from LS-Prepost in the provided workshop folder. Because it is a step file, you can see that the multiple objects are imported together.

- **Import → Step → select CAD model**
- **Step Read Options → setup (see image)**



STEP Read Options

● Check Unit

○ Specify Unit m

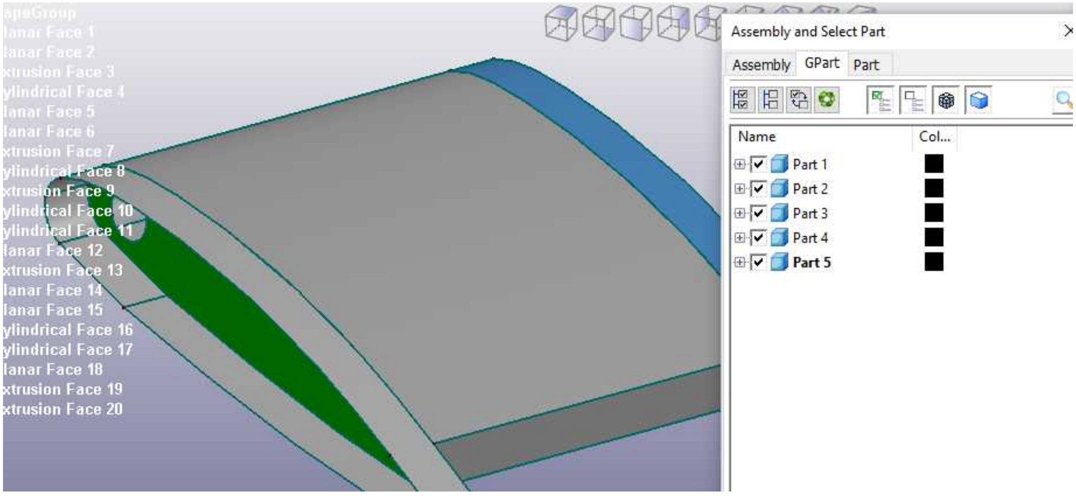
○ Scale Factor 1.0

☐ Don't pop-up any more.

▲ Step Read Option and imported parts

Let's take a look at the imported CAD model by looking around in LS-Prepost. The model is shaped like a shell of the wing box of the aircraft main wing. If you select the **Assembly** and select **Part**, the imported model consists of 5 parts as shown below.

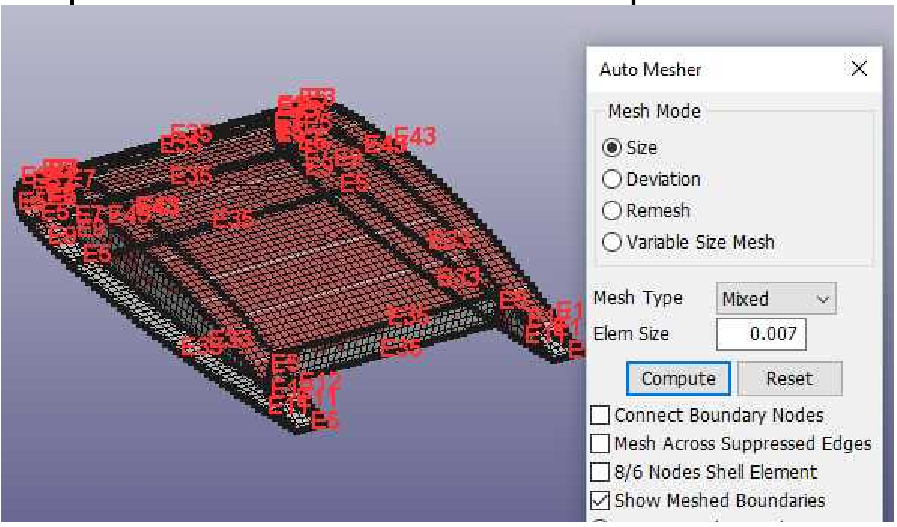
- **Model and Part → Assembly and Select Part → select all**



▲ Imported geometry part

Let's create elements in this part. If you create shell elements using the **Automesh** option, a mesh will be created as shown below.

- **Element and Mesh → Auto Mesh → Select Geom | Area → select all by drag mouse → Auto Mesher | Compute → element size= 0.007 → Mesh → Accept**


Auto Mesher

Mesh Mode

● Size
○ Deviation
○ Remesh
○ Variable Size Mesh

Mesh Type: Mixed

Elem Size: 0.007

Compute | Reset

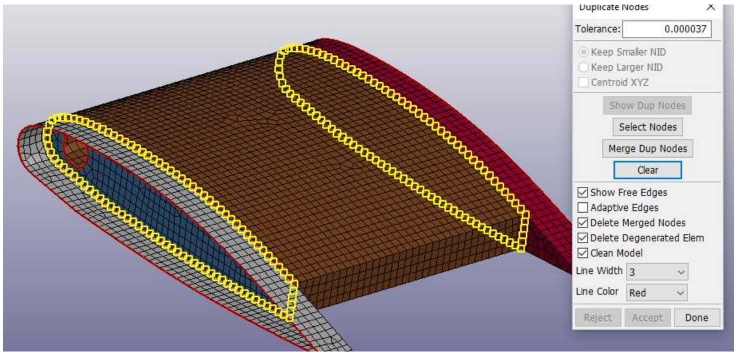
☐ Connect Boundary Nodes
☐ Mesh Across Suppressed Edges
☐ 8/6 Nodes Shell Element
☑ Show Meshed Boundaries

▲ Automesh setup

Check for free edges and delete duplicate nodes

Let's check that there are no elements that are not connected to each other. If you check the free edge and the duplicated nodes which are two nodes in one location, you will see duplicate nodes at the multiple edges as shown below.

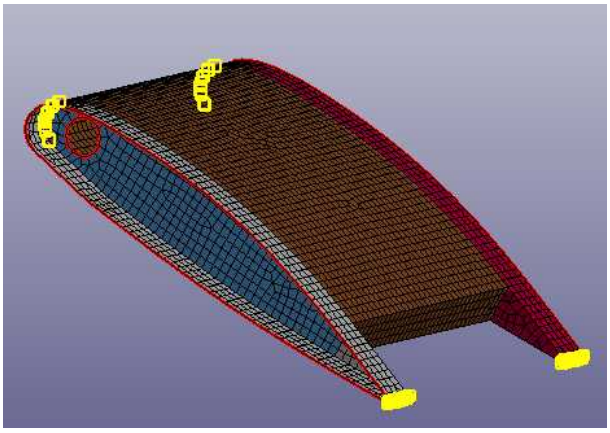
- **Element and Mesh → Edge face → free edge**
- **→ Duplicate Nodes → Show DUP nodes**
- **→ Merge DUP nodes to delete duplicated nodes**



Tolerance: 0.000037
● Keep Smaller NID
○ Keep Larger NID
☐ Centroid XYZ
☑ Show Free Edges
☐ Adaptive Edges
☑ Delete Merged Nodes
☑ Delete Degenerated Elem
☑ Clean Model
Line Width 3
Line Color Red

▲ Duplicated nodes distribution automesh setup

If you check the free edge again, some parts will remain as shown below. After increasing Tolerance to 0.004, click on '**Merge DUP nodes'** which will remove all duplicated nodes.

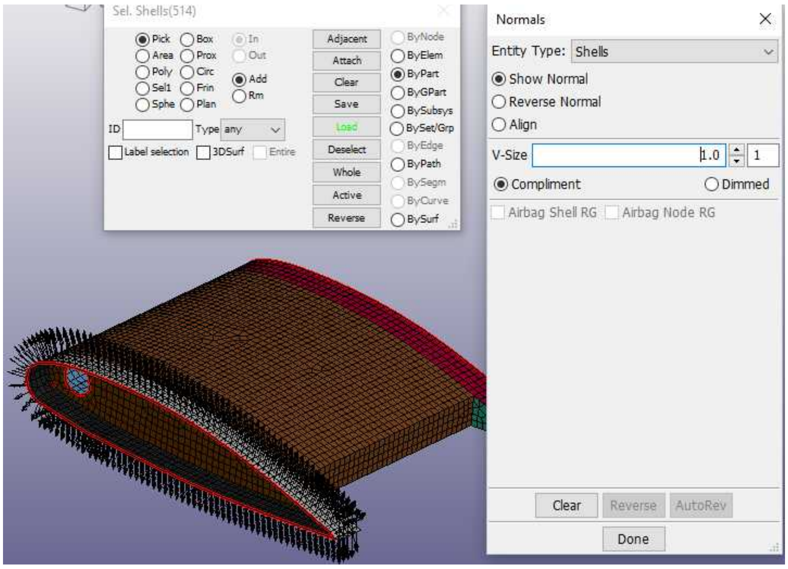


▲ Remaining Duplicated nodes

Shell Normal

It is important to set the proper Normal direction in the shell element. If the normal direction of shell elements are mutually dissimilar to each other, they should be matched with each other through Reverse Normal using the Normals function as shown below :

- **Element and Mesh → Normals | Shell | Show Normal → Sel. Shell | Pick | By Part → pick a part to check Normal direction**



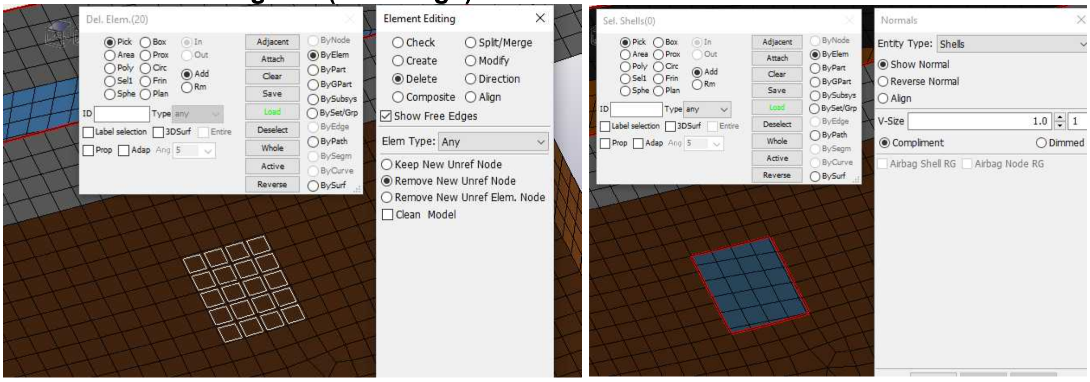
Entity Type: Shells
● Show Normal
○ Reverse Normal
○ Align
V-Size 1.0 1
● Compliment ○ Dimmed
☐ Airbag Shell RG ☐ Airbag Node RG

▲ Checking Shell Normal direction

Modify elements

The next step is to modify the element to create the window for inspection on the Wing Skin.

- **Element and Mesh → Element editing | Delete → Sel. Shell | By element | pick → select elements in wing skin(see image)**



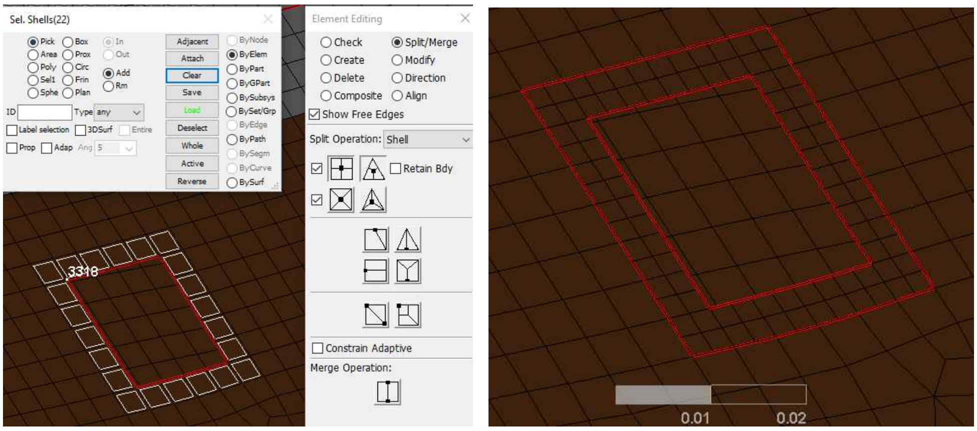
Element Editing:
○ Check ○ Split/Merge
○ Create ○ Modify
● Delete ○ Direction
○ Composite ○ Align
☑ Show Free Edges
Elem Type: Any
○ Keep New Unref Node
● Remove New Unref Node
○ Remove New Unref Elem. Node
☐ Clean Model

Normals:
Entity Type: Shells
● Show Normal
○ Reverse Normal
○ Align
V-Size 1.0 1
● Compliment ○ Dimmed
☐ Airbag Shell RG ☐ Airbag Node RG

▲ Element removal at the bottom of the wing skin

The inspection window area is the stress concentration area, so let's divide the surrounding elements into more subdivisions. If you select after ‘accept’ as shown in the figure below, you can see that the element around the check window is smaller.

- **Element and Mesh → Element editing | Split → Sel. Shell | By element | pick → select element around window → Accept → Done**



Element Editing

● Split/Merge

☑ Show Free Edges

Split Operation: Shell

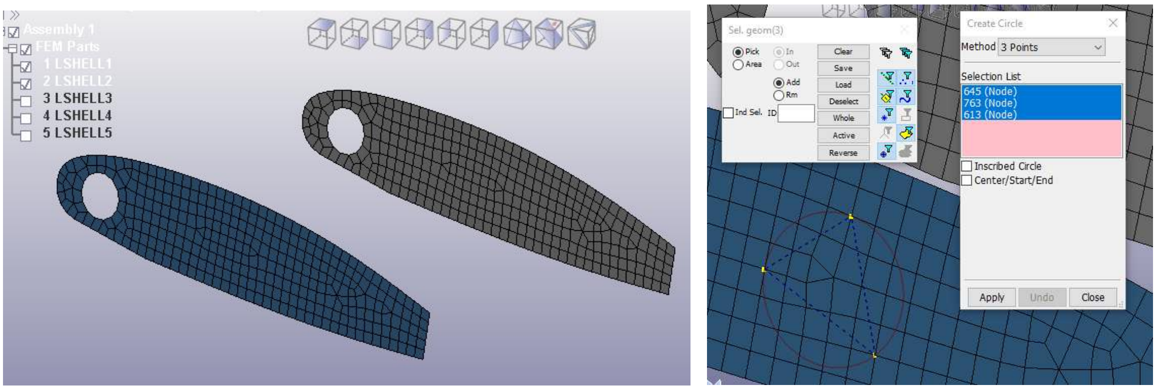
☐ Retain Bdy

☐ Constrain Adaptive

▲ Split of elements around the inspection window

This time, we will create a hole through the trim on the side wing rib. After activating only parts 1 and 2 as shown below, draw holes on the surface of part 1 referring to the picture below.

- **Geometry → Curve → Circle | Method=3 points → select node(see image) → Apply → Close Element tools → Part trim → Part = 1, 2 | Curve = select created Circle → Trim method = Closest 3D proj. dir → Trim → Accept**
- **→Element editing → delete → select internal element in circle → delete → done**



Create Circle

Method 3 Points

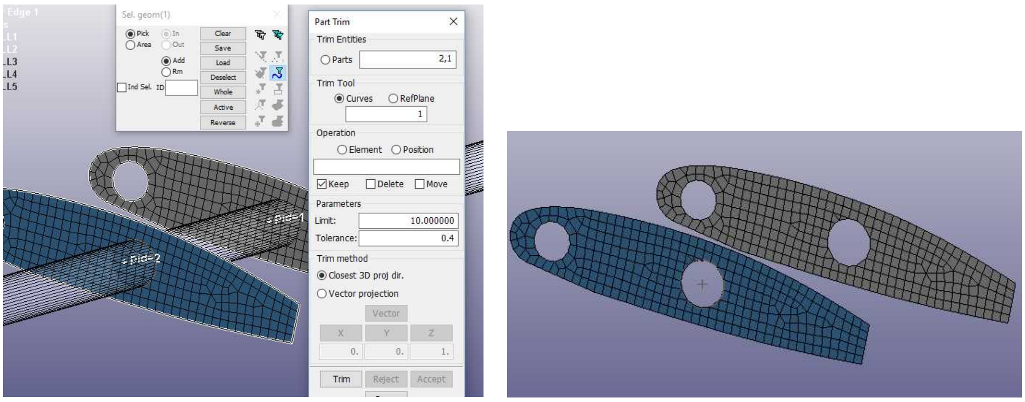
Selection List

645 (Node)
763 (Node)
613 (Node)

☐ Inscribed Circle

☐ Center/Start/End

▲ Draw Hole on Wing Rib



Part Trim

Trim Entities

○ Parts 2,1

Trim Tool

● Curves ○ RefPlane 1

Operation

○ Element ○ Position

☑ Keep ☐ Delete ☐ Move

Parameters

Limit: 10.000000

Tolerance: 0.4

Trim method

● Closest 3D proj dir.

○ Vector projection

X 0. Y 0. Z 1.

▲ Trim in Wing Rib

So far from this chapter, you have practiced the process of importing CAD models and creating elements using some features of LS-Prepost. The mesh generation procedure in the LS-Dyna analysis model is the most frequently used method. There will not be many cases where modeling and meshing are directly done in LS-prepost. In addition, Hypermesh or Ansys provides a powerful function that supports meshing and later helps to create a better LS-DYNA model.

W4.1

Workshop 4.1
Create Part from Meshed Model

In this exercise, we will import the mesh model which is created by an external pre-processor into LS-Prepost and then create detailed parts, check the quality of the mesh and make corrections. In any LS-DYNA analysis work, it is often necessary to reconstruct the LS-DYNA model from the externally provided model.

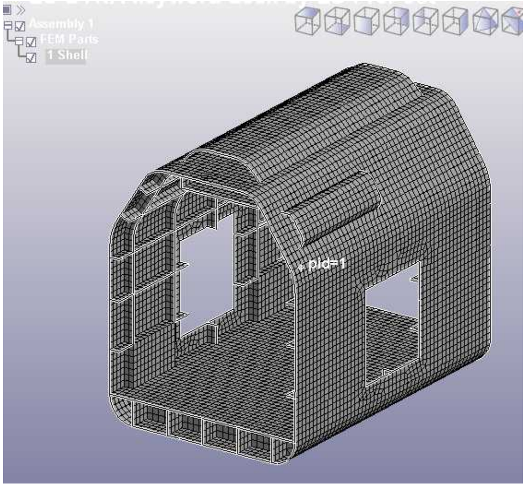
Required model

- W41_PartGeneration.bdf: Initial state model
- W41_FuselageModel.k: Final state model

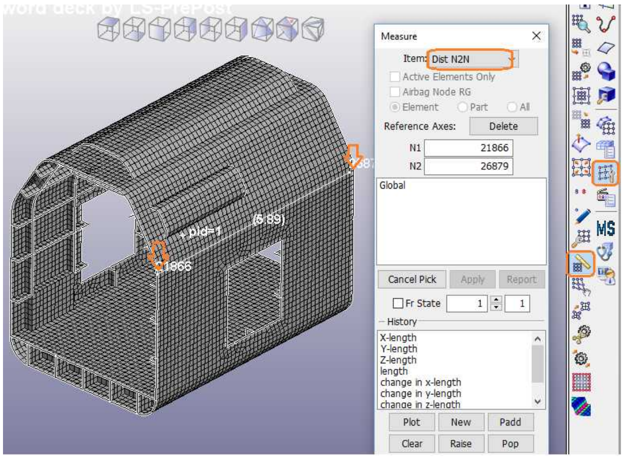
Import CAD model

Let's import the CAD model named 'W41_PartGeneration.bdf' from the provided workshop folder. As shown in the figure below, a single helicopter fuselage part consisting of Shell elements is imported. If you measure the length with the **Measure** function, you can see its length is about 6m. Therefore, the unit system adopts **MKS** [m, kg, sec].

- **File → Import → Nastran → W41_PartGeneration.bdf**
- **Element tools → Measure → Dist_N2N → Measure the length by selecting both nodes**



▲ Imported model



Item: Dist N2N

☐ Active Elements Only

☐ Airbag Node RG

● Element ○ Part ○ All

Reference Axes:

N1 21866

N2 26879

☐ Fr State 1 1

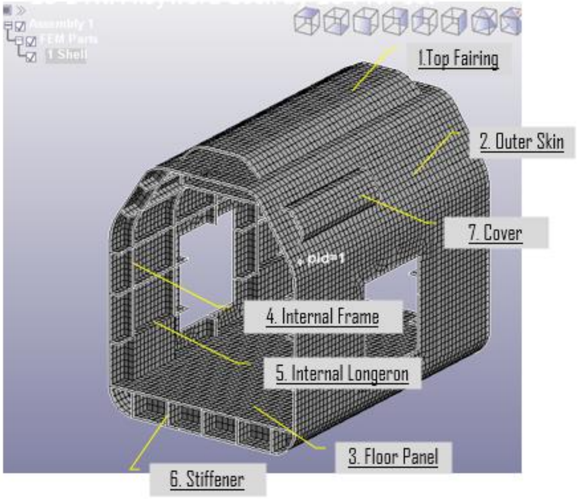
X-length
Y-length
Z-length
length
change in x-length
change in y-length
change in z-length

▲ Measure function to determine the length of the fuselage

Rebuild Meshed Part

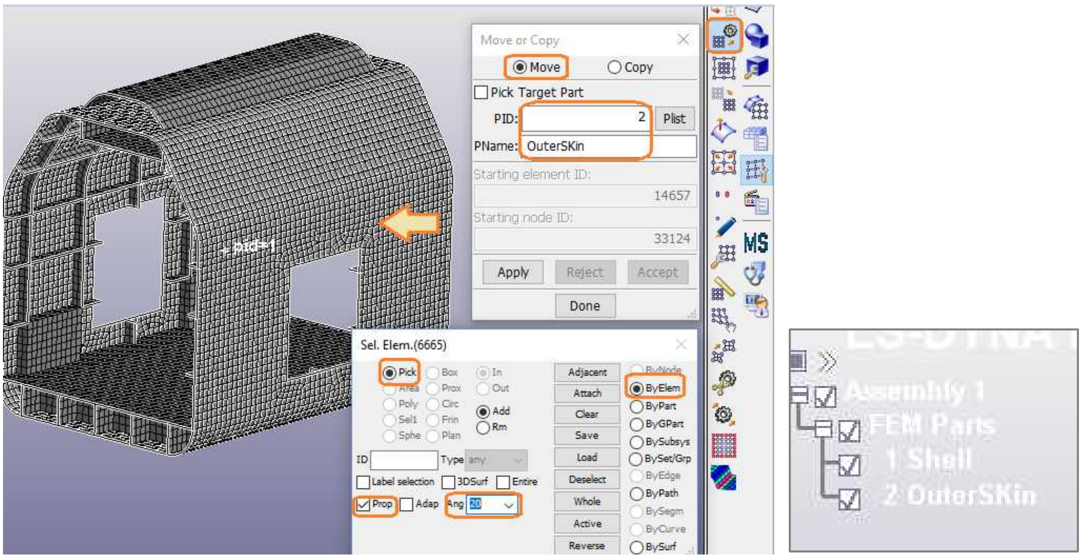
In this way, we will rebuild the mesh model consisting of several parts by segregating the sections of each component/mesh. Helicopter fuselage structures will be divided into the following parts:

- Part 1: Top Fairing
- Part 2: Outer Skin
- Part 3: Floor Panel
- Part 4: Internal Frame
- Part 5: Internal Longeron
- Part 6: Bottom Stiffener
- Part 7: Exhaust Cover



Separate elements of each individual component parts using **Move or copy** in the element editing functions. The external skin is selected and created as a new **Part2** using the following functions:

- **Element tools → Move or copy → PID=2, outerskin → Sel Element → Pick | Add | Prop | Ang=20 | select external skin → Apply → Accept →Done**



Move or Copy

● Move ○ Copy

☐ Pick Target Part

PID: 2

PName: OuterSKin

Starting element ID: 14657

Starting node ID: 33124

Sel. Elem.(6665)

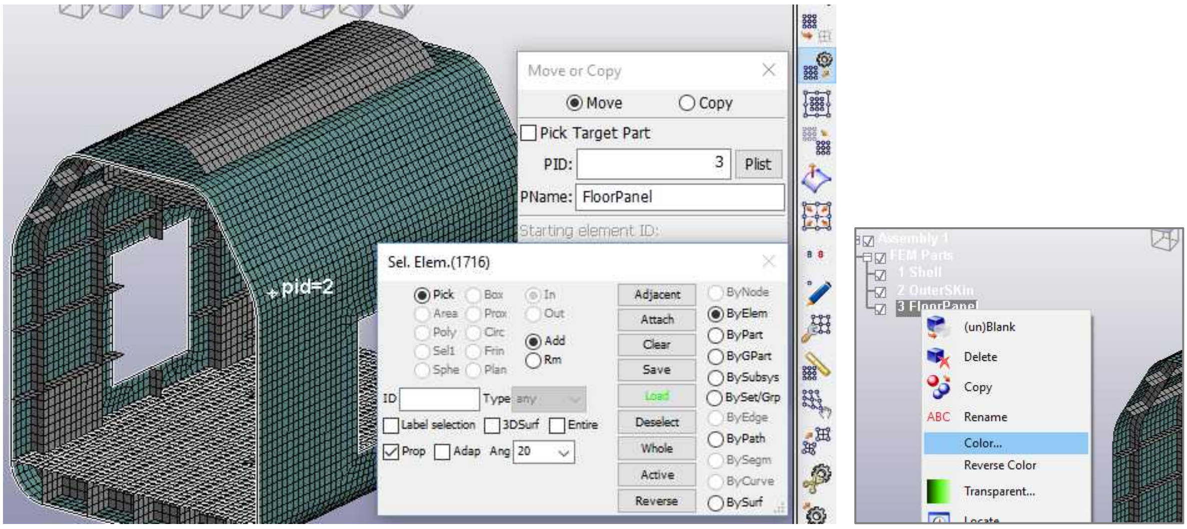
● Pick ● Add ● ByElem

☑ Prop ☐ Adap Ang 20

▲ Creating Part2 Outerskin Part Using Move or Copy Function

This new part can be checked in the model tree on the left side of the screen. Let's create the rest of the remaining components/mesh as new parts. Let's change the color by right-clicking on the model tree to create a new part.

- **Element tools → Move or copy → PID=3, FloorPanel → Sel Element → Pick | Add | Prop | Ang=20 | select internal floor → Apply → Accept →Done**



Move or Copy

● Move ○ Copy

☐ Pick Target Part

PID: 3

PName: FloorPanel

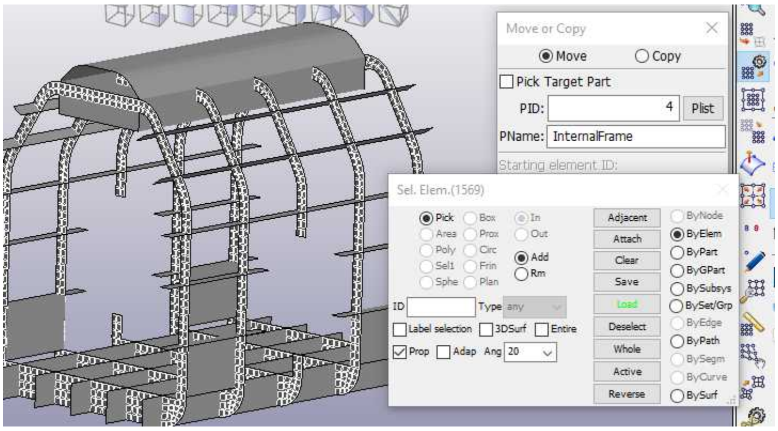
Sel. Elem.(1716)

● Pick ● Add ● ByElem

☑ Prop ☐ Adap Ang 20

▲ Creating Part3 FloorPanel Part Using Move or Copy Function

In the same way, let's divide Part4 and Part5 into new parts by referring to the following image. To make it easier to select elements, previously created parts in the model tree are removed/ hidden from the screen, making it easier to select the remaining parts. There are various ways to select elements, such as **Pick**, **Area** (drag with mouse), **Poly** (polygon selection), so apply these options several times to achieve required action. When creating Part5, the element separated by Cover can be removed from the selection by the **Rm** (remove) function as shown below.



Move or Copy

● Move ○ Copy

☐ Pick Target Part

PID: 4

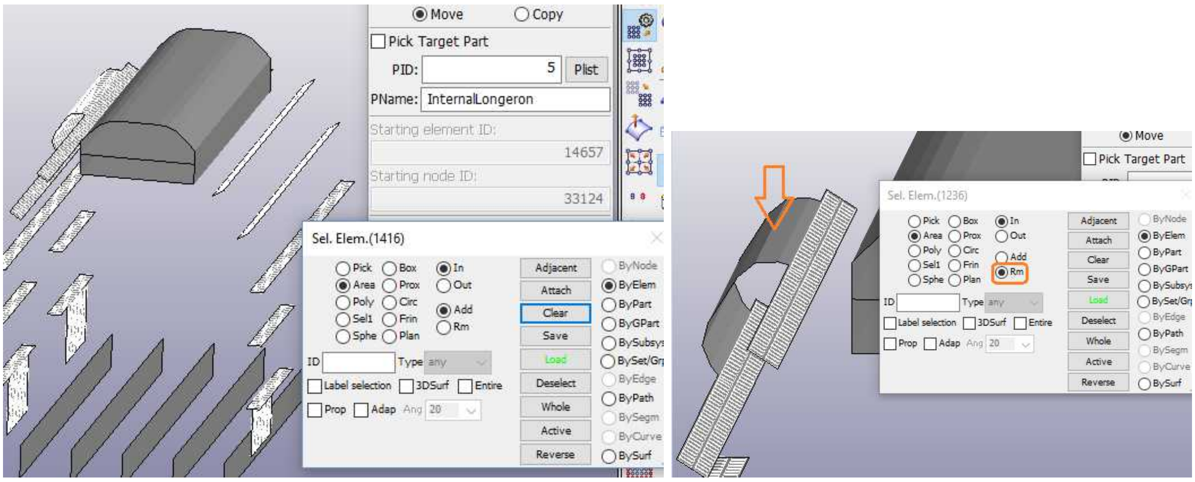
PName: InternalFrame

Starting element ID:

Sel. Elem.(1569)

● Pick, ● Add, ● ByElem, ☑ Prop, Ang 20

▲ Create Part4 Internal Frame



● Move ○ Copy

☐ Pick Target Part

PID: 5

PName: InternalLongeron

Starting element ID: 14657

Starting node ID: 33124

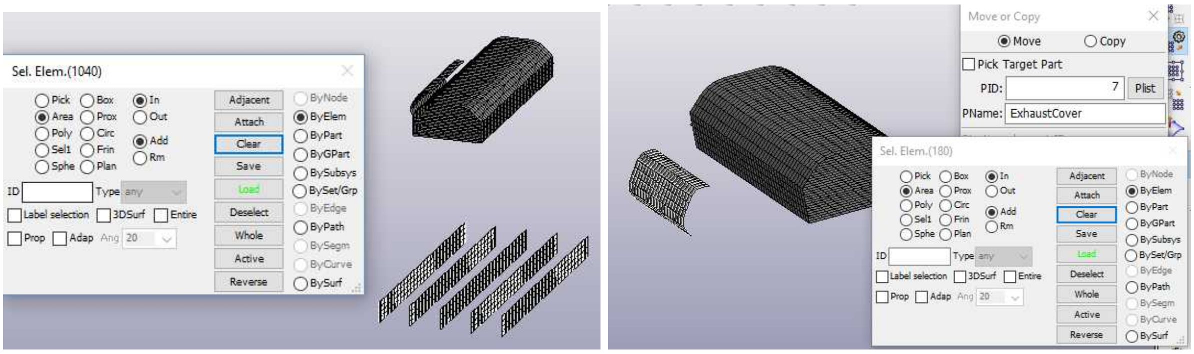
Sel. Elem.(1416)

● Area, ● In, ● Add, ● ByElem, Ang 20

Sel. Elem.(1236)

● Area, ● In, ● Rm, ● ByElem, Ang 20

▲ Create Part5 Internal Longeron



Sel. Elem.(1040)

● Area, ● In, ● Add, ● ByElem, Ang 20

Move or Copy

● Move ○ Copy

☐ Pick Target Part

PID: 7

PName: ExhaustCover

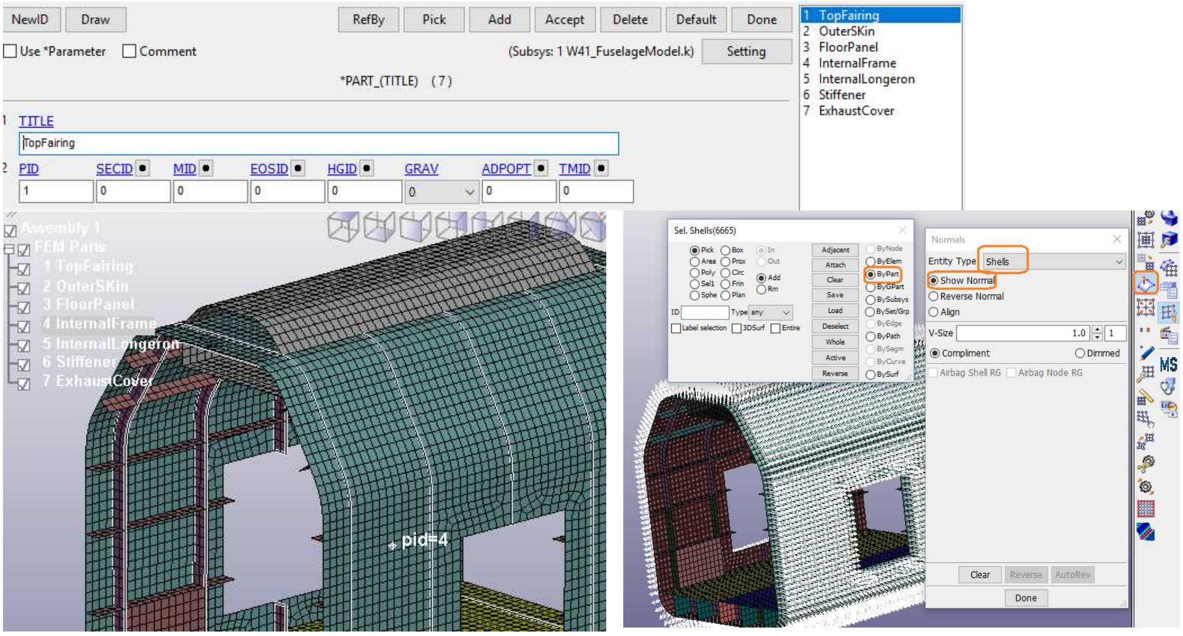
Sel. Elem.(180)

● Area, ● In, ● Add, ● ByElem, Ang 20

▲ Create Part6~8

In this way, the model was reconstructed into seven parts as shown below. If you select **Part** in **Keyword Manager**, you can see seven * **Parts** created as shown below. After creating **SECID** and **MID** for each part, assign them to each part. Since this is a shell element model, make sure that the shell normal direction matches. The white arrow highlights the Normal direction. The surface region where collision or contact occurs should match the normal direction, so match it using **Reverse** or **Align** function.

- **Element tools → Shell Normal → Type=Shell → Show Normal → Check Normal direction and use Reverse to modify**



*PART_(TITLE) (7)

1 TITLE: TopFairing

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
1	0	0	0	0	0	0	0

1 TopFairing
2 OuterSKin
3 FloorPanel
4 InternalFrame
5 InternalLongeron
6 Stiffener
7 ExhaustCover

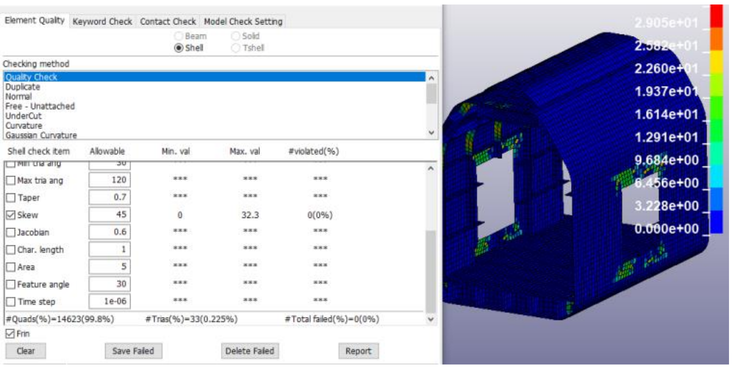
Entity Type: Shells
● Show Normal
○ Reverse Normal
○ Align
V-Size: 1.0
● Compliment ○ Dimmed

▲ Check the normal shell

Edit Mesh Model

If the working model is well generated, click the **Model Check** button of **Keyword Manager** to check various states such as mesh quality and contact condition. This is a good practice step to check the area where the skew angle is less than 45 degrees through mesh quality check.

- **Keyword Manager → Model Check → Method | Quality Check | Skew | Allowable=45 → Check**



Shell check item	Allowable	Min. val	Max. val	#violated(%)
☐ Min tria ang	30			
☐ Max tria ang	120	***	***	***
☐ Taper	0.7	***	***	***
☑ Skew	45	0	32.3	0(0%)
☐ Jacobian	0.6	***	***	***
☐ Char. length	1	***	***	***
☐ Area	5	***	***	***
☐ Feature angle	30	***	***	***
☐ Time step	1e-06	***	***	***

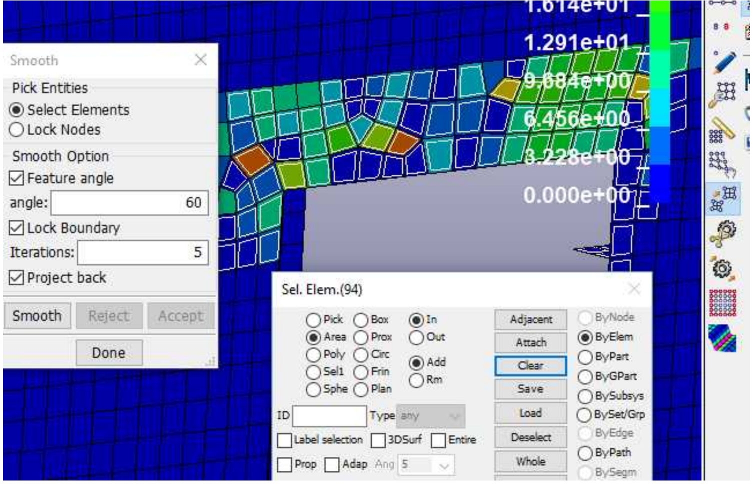
#Quads(%)=14623(99.8%) #Trias(%)=33(0.225%) #Total failed(%)=0(0%)

☑ Frin

▲ Mesh quality check

The next step is to apply the function **Smooth**, a feature that selects and improves elements that are somewhat of poor quality. In the above model, if you select the part with insufficient skew state and select the **Smooth** function, you can partially improve the quality of the element. The element quality of this training model is generally acceptable, so it does not affect the analysis results. However, if the quality of the element is poor, it takes a long time to analyze and the possibility of getting an error is high. Therefore, this **Model Check** procedure should be applied to important models.

- **Element tools → Smooth → Select elements, select poor quality elements → Angle=60, Iteration=5, Smooth → Accept**

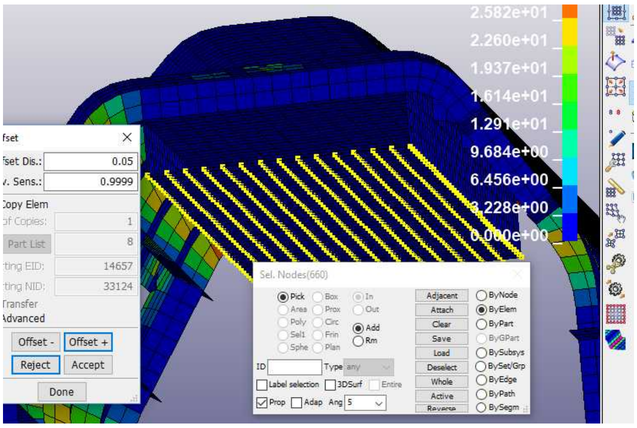


● Select Elements
○ Lock Nodes
☑ Feature angle
angle: 60
☑ Lock Boundary
Iterations: 5
☑ Project back

▲ Smooth application to improve mesh state

The next step is the **offset** function, which moves a specific set of elements to increase the depth at the bottom of the top fairing. You can move it long or small in the vertical direction of the selected element set.

- **Element tools → Offset → Distance=0.08, Sensitivity=0.99 → select Fairing lower element set → Offset + / - adjustment → Accept**

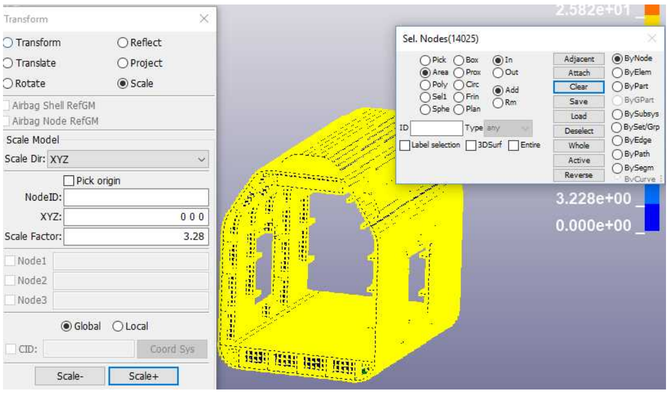


fset Dis.: 0.05
v. Sens.: 0.9999
of Copies: 1
Part List: 8
ting EID: 14657
ting NID: 33124

▲ offset setting to change element set position

Lastly, the unit consistency method is to be applied when the model unit needs to be changed due to the request of the company or the drawing requirement. Currently, MKS is a unit system and the distance unit is a model composed of meter units. There are several ways to convert meter into feet. The first is to specify the **Scale Factor** when you export the model to geometry and re-import it again. In this case, there is a drawback of having to mesh again. The second is to use the **Element tools → Transform → Scale** function to change the unit system while maintaining the current mesh state. Since 1 meter is 3.28 feet, entering **Scale Factor** = 3.28 will change the model geometry unit from meters to feet. **Transform** function is a useful function that can handle various parts such as moving, rotating, reducing and enlarging parts.

- **Element tools → Transform → Scale → select whole part → scale dir=XYZ | Origin XYZ [0 0 0] | Scale Factor=3.28 → Scale +/- adjustment → Accept**



Scale Dir: XYZ
XYZ: 0 0 0
Scale Factor: 3.28

▲ Scale application to change the unit system

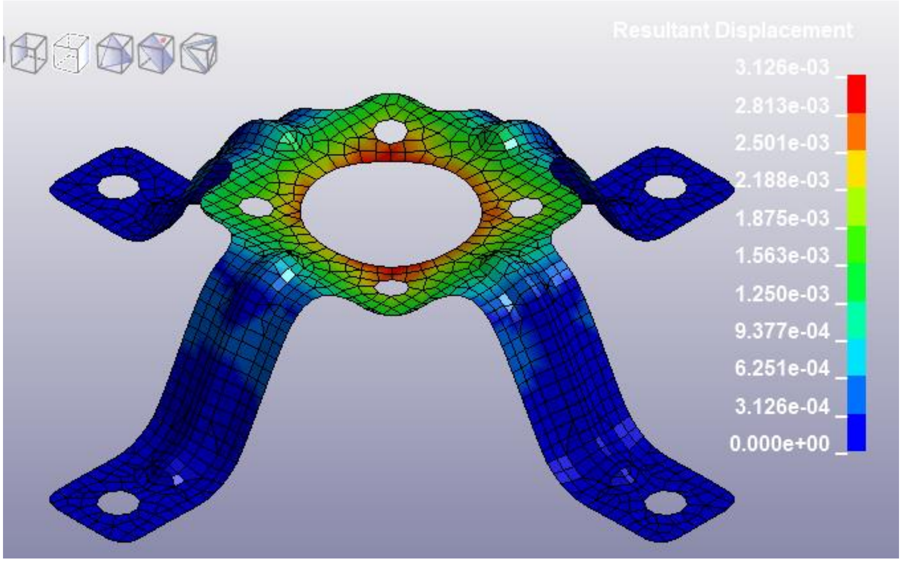
So far, we have practiced the modeling process such as segregating one mesh model into various parts to meet the purpose of analysis, modifying elements and changing the unit system. This process is a necessary procedure for FE modeling management in the analysis practice, so please use the various functions in LS-Prepost. There are dozens of ways to create one model and applying the fastest and most effective way is the analyst's ability. Keep in mind that the analyst’s capability can build up through many trials and experiences.

Part B

Beginner Level

In this chapter, you will practice the basics of LS-DYNA analysis. You will practice all the steps from the geometry creation and mesh generation to the LS-DYNA keyword generation, analysis, and result review through simple examples. Analysis models are prepared with multi-disciplinary fields and readers can also go through the basic theory explanation.

Chapter 5. Static Analysis of 3D Bracket



Workshop Objectives

✓ **Mesh Creating a Mesh by Importing a CAD Mode**

✓ **Get familiar with the LS-Prepost workplace**

✓ **Practice basic 3D modeling methods**

✓ **Learn how to apply static load**

Duration: 40 min

Application: Static Load, Explicit, Structural

Model Files

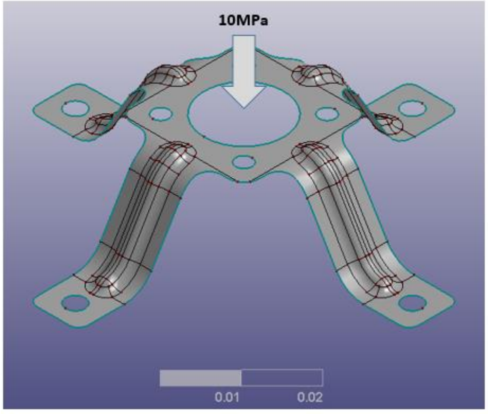
- **W5_Geometry_Shell.iges:** Geometry model
- **W5_SETUP_BracketModel.k:** Final state model

W5

Workshop 5.
Static Analysis of 3D Bracket

Introduction

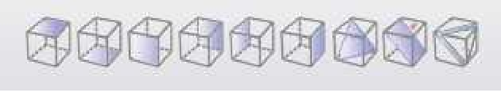
Readers who are new to LS-DYNA will spend a lot of time getting used to the modeling structure. Recently, other programs are designed intuitively through many developments in the GUI (Graphic User Interface) for the user's convenience, so that the learning time can be effectively reduced, whereas LS-DYNA requires an understanding of the Keyword structure for effective modeling. In this lesson, we will use LS-Prepost to create and analyze the 3D bracket structure, which is generally learned in the first stage of the FEM program. In this exercise, you will be familiar with the LS-Prepost environment and understand the process of reviewing the results. There is no need for prior knowledge or additional learning required and this is an example that can be completed by just following step by step. The model covered in the workshop will be a 3D bracket that imports CAD model (iges format) into LS-Prepost to create a mesh and create necessary input keywords. 4 bolts mount of the 3D bracket are fixed and 10MPa load is applied to the upper surface. The material is aluminum and the unit system is the MKS system.



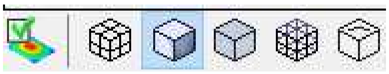
▲ 3D Bracket Model

Importing Geometry

If you import the provided geometry model (W5_geometry_shell.iges), you will see the following model on screen. Let's see how the model looks by clicking on the **View** icon at the top. As you get used to the **View** icon, you can speed up the modeling process. The model is the shell-type bracket and having a total size of about 70mm. **Measure ruler** is useful to check the macro model size easily, so activate and proceed to model. When **Measure ruler** is enabled in the Display option, its icon is displayed at the bottom of the viewport.

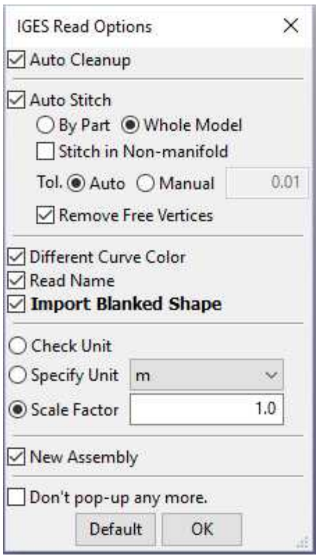


◀view icon



◀display option

- **File → Import → Iges file → W5_geometry_shell.iges, default in option**



IGES Read Options

☑ Auto Cleanup

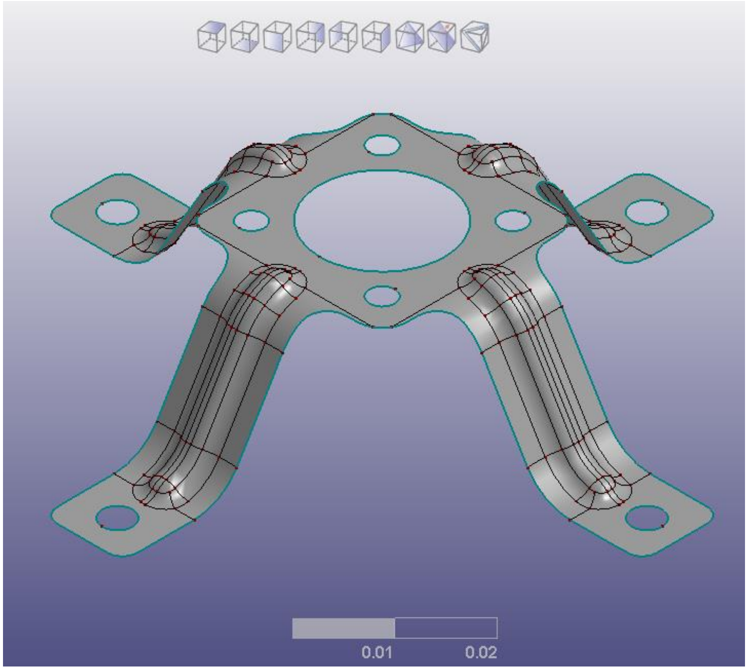
☑ Auto Stitch
  ○ By Part ● Whole Model
  ☐ Stitch in Non-manifold
  Tol. ● Auto ○ Manual 0.01
  ☑ Remove Free Vertices

☑ Different Curve Color
☑ Read Name
☑ **Import Blanked Shape**

○ Check Unit
○ Specify Unit m
● Scale Factor 1.0

☑ New Assembly

☐ Don't pop-up any more.

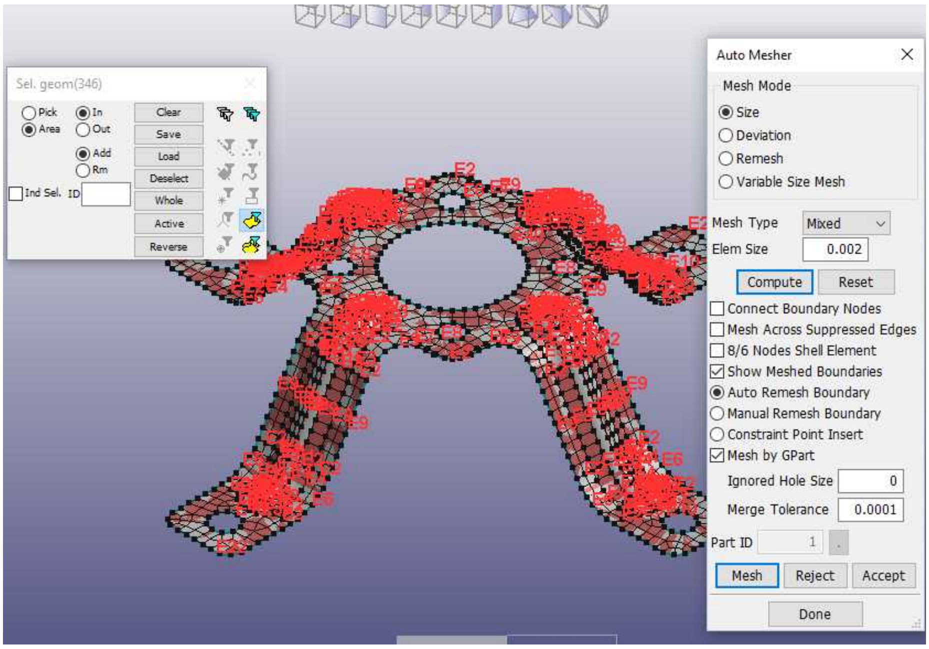


▲ Imported Geometry model

Mesh Generation

Element creation is done using the **Auto Mesher**. If you calculate the element size automatically with the auto mesher, a 2.4mm mesh will be created as shown below. If you look closely, you will see that the rectangular and triangular elements are properly combined to bound well along the geometry plane.

- **Element and Mesh → Auto Mesher → Sel. Geom | Area, In, add, drag all elements→ option | size, compute → Mesh → Accept**



Sel. geom(346)

○ Pick ● In
● Area ○ Out
● Add ○ Rm
☐ Ind Sel. ID

Auto Mesher

Mesh Mode
● Size
○ Deviation
○ Remesh
○ Variable Size Mesh

Mesh Type: Mixed
Elem Size: 0.002

☐ Connect Boundary Nodes
☐ Mesh Across Suppressed Edges
☐ 8/6 Nodes Shell Element
☑ Show Meshed Boundaries
● Auto Remesh Boundary
○ Manual Remesh Boundary
○ Constraint Point Insert
☑ Mesh by GPart

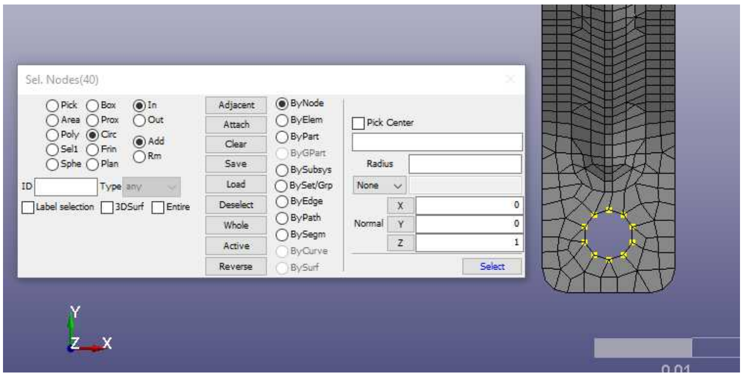
Ignored Hole Size: 0
Merge Tolerance: 0.0001
Part ID: 1

▲ Generated Mesh Model

Boundary Condition and Load Setup

We shall fix all four holes on the bottom of the bracket and apply a pressure of 10MPa on to the top surface. First, let's view the model by selecting the top view from the view icons. Then, select Node inside the hole.

- **Model and Part → Create Entity → Boundary | SPC | Cre → Sel. Node | Cir | by node, select nodes around hole(see image) → check all x,y,z,rx,ry,rz → Apply**



Sel. Nodes(40)

○ Pick ○ Box ● In
○ Area ○ Prox ○ Out
○ Poly ● Circ ● Add
○ Sel1 ○ Frin ○ Rm
○ Sphe ○ Plan

ID Type: any
☐ Label selection ☐ 3DSurf ☐ Entire

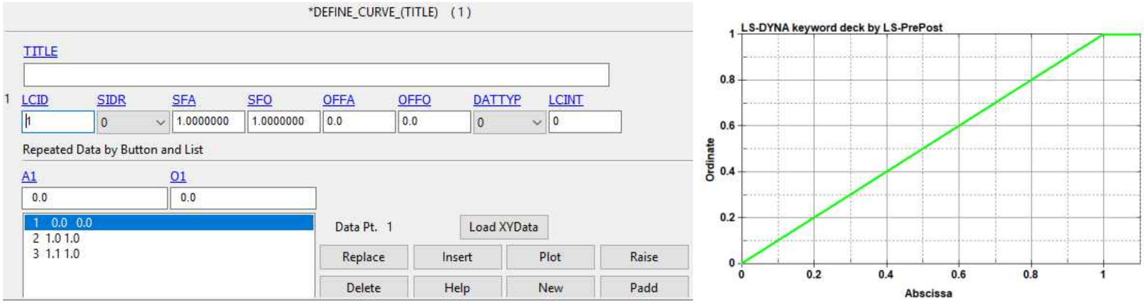
● ByNode
○ ByElem
○ ByPart
○ ByGPart
○ BySubsys
○ BySet/Grp
○ ByEdge
○ ByPath
○ BySegm
○ ByCurve
○ BySurf

☐ Pick Center
Radius
None
Normal X: 0
Normal Y: 0
Normal Z: 1

▲ Boundary condition setup

To apply the load, create a curve represents the load profile. Since the step time of the analysis proceeds for 1 second, enter a load period of 1.1 seconds so that the magnitude of the load starts at 0 and becomes 1 at maximum in 1 second and then is maintained for a certain period of time.

- **Keyword Manager → Define Curve → LCID=1, input values in A1 and O1 (0,0 insert | 1,0 insert | 1.1, 0 insert → accept → plot**


*DEFINE_CURVE_(TITLE) (1)

TITLE

LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	LCINT
1	0	1.0000000	1.0000000	0.0	0.0	0	0

Repeated Data by Button and List

A1: 0.0
O1: 0.0

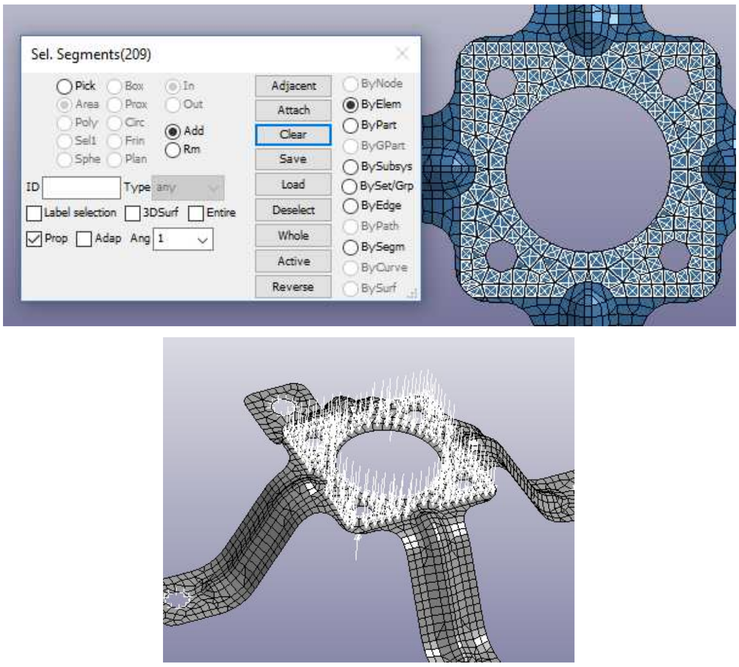
1 0.0 0.0
2 1.0 1.0
3 1.1 1.0

Data Pt. 1

▲ Define Curve input and generated load graph

- **Model and Part → Create Entity → Load Segment | Cre → Sel. SEg | Pick | Prop, angle =1, select top surface → LCID=1, SF=10e6 → apply**

This confirms that the boundary condition and the load condition are input. When modeling shell elements, sometimes the normal direction of shell elements is inconsistent, so use the **Element tools → Normal function** to check that the normal direction vectors for all elements match in the same direction.


Sel. Segments(209)

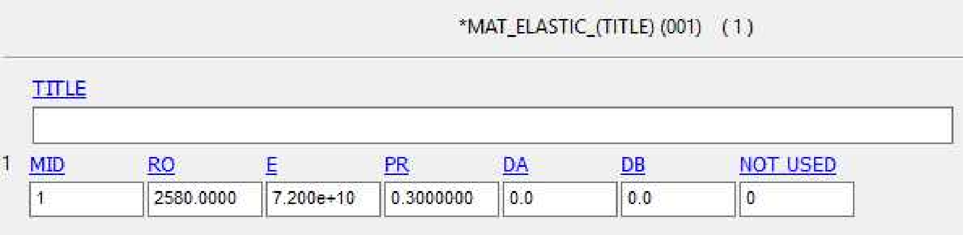
○ Pick ● Add ● ByElem ☑ Prop Ang 1

▲ Loading condition setup

Material Property and Section Property

Assuming that the bracket to be made of aluminum, enter the property of Aluminum using the keyword **MAT001_Elastic** as shown below. Since it is a meter unit system, input density 2580kg/m^3, Young's Modulus 72GPa, Poisson's ratio = 0.3.

- **Keyword Manager → MAT → 001_Elastic | input Ro, E, PR(see image)**



*MAT_ELASTIC_(TITLE) (001) (1)

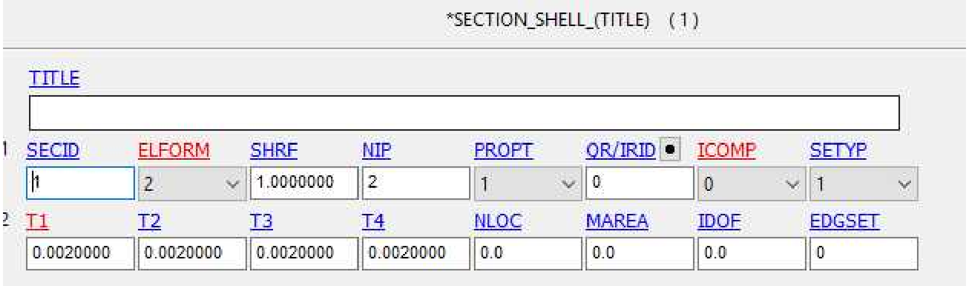
TITLE

	MID	RO	E	PR	DA	DB	NOT USED
1	1	2580.0000	7.200e+10	0.3000000	0.0	0.0	0

▲ Material Property input

To input the properties of the shell element, enter the section properties referring to the image below. Since the shell thickness is 2mm, enter 0.002 in T1.

- **Keyword Manager → SECTION → Shell | Elform =2, NIP=2, T1=0.002**



*SECTION_SHELL_(TITLE) (1)

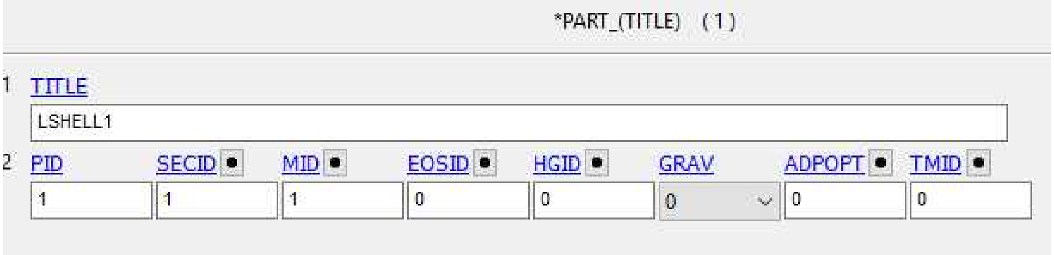
TITLE

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
	1	2	1.0000000	2	1	0	0	1
2	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
	0.0020000	0.0020000	0.0020000	0.0020000	0.0	0.0	0.0	0

▲ Section Property input

After entering the material properties and element property values, assign these created material properties created into the part.

- **Keyword Manager → Part → PID = 1, SECID = 1, MID = 1**



*PART_(TITLE) (1)

1 TITLE

LSHELL1

	PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
2	1	1	1	0	0	0	0	0

▲ Part Data input

Analysis Condition

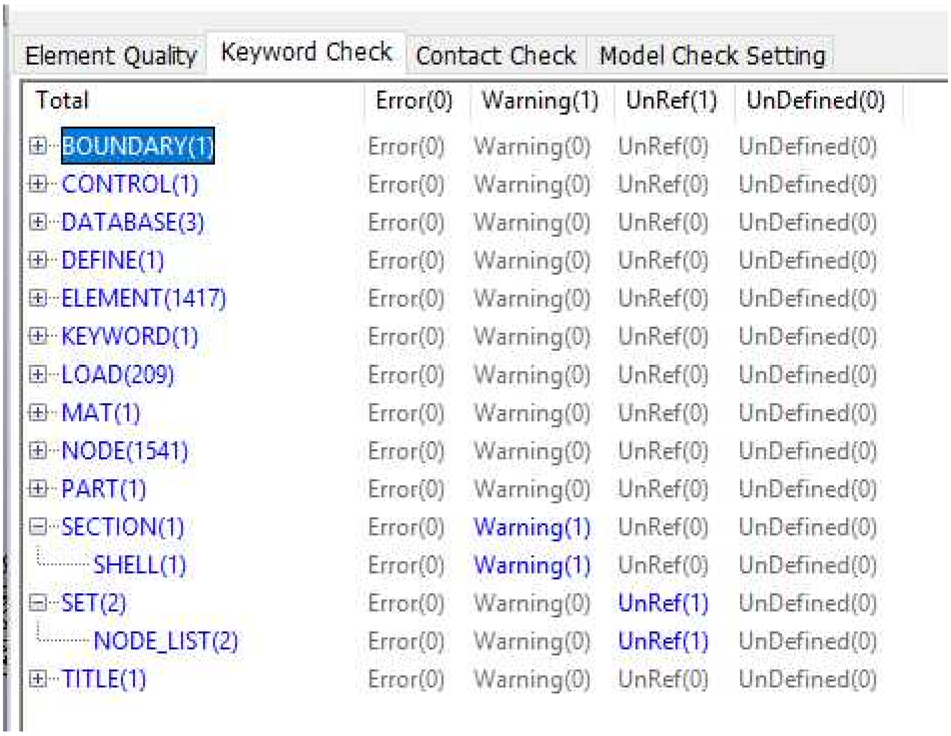
This step is to setup various analysis conditions such as analysis time and result output interval request for analysis.

- **Keyword Manager → Control → Termination | ENDTIM = 1 second input**
- **Keyword Manager → Database → Binary_D3plot = 0.1**

- **Keyword Manager → Database → ASCII_Option | Default DT = 0.02 input, select GLSTAT, MATSUM**

This is the process of generating keywords for the analysis model. To check if all the necessary parts of the model are included, use **Model checking** to check for missing elements. If no error appears, it means that all the necessary elements are included in the analysis model. In this model, two warnings come up and double-clicking them individually confirms detailed information, which can be useful when debugging. In this model, the contents of the selected node and the shell thickness warning during the modeling process are irrelevant to the analysis results. Let's compare the generated keyword with the "***W5_SETUP_Bracket Model.k***" keyword provided in the workshop folder. The model generated by the exercise might be slightly different in terms of node and element count.

- **Upper Main Menu Application → Model Checking → General checking → Keyword check**

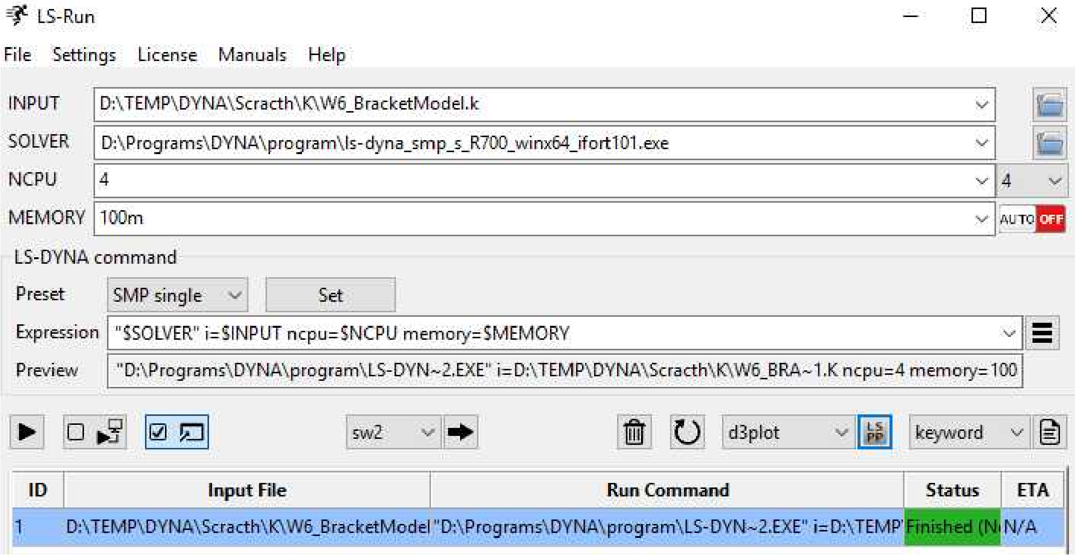


Total	Error(0)	Warning(1)	UnRef(1)	UnDefined(0)
⊞ BOUNDARY(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ CONTROL(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ DATABASE(3)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ DEFINE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ ELEMENT(1417)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ KEYWORD(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ LOAD(209)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ MAT(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ NODE(1541)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊞ PART(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)
⊟ SECTION(1)	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)
SHELL(1)	Error(0)	Warning(1)	UnRef(0)	UnDefined(0)
⊟ SET(2)	Error(0)	Warning(0)	UnRef(1)	UnDefined(0)
NODE_LIST(2)	Error(0)	Warning(0)	UnRef(1)	UnDefined(0)
⊞ TITLE(1)	Error(0)	Warning(0)	UnRef(0)	UnDefined(0)

▲ Keyword checking input

Job Execution

There are many ways to perform an analysis. There is a method of executing a command directly in the command window. If you are using ANSYS, you can use the **command window** provided by ANSYS or use **LS-DYNA Manager**. In this workshop, we will use a useful job execution feature provided by LS-Prepost. If you click **File** → **Run LS-DYNA**, a dialog box for solving is displayed below. Here various functions are provided. They are model file, storage location, solver selection, CPU number, and memory allocation options. Selecting the triangle below gives a brief overview of the analysis process. Also, the result file is directly linked and the keyword can be checked directly with the text editor, so use it frequently to familiarize with the working environment.



INPUT: D:\TEMP\DYNA\Scracth\K\W6_BracketModel.k
SOLVER: D:\Programs\DYNA\program\ls-dyna_smp_s_R700_winx64_ifort101.exe
NCPU: 4 | 4
MEMORY: 100m

LS-DYNA command
Preset: SMP single
Expression: "$SOLVER" i=$INPUT ncpu=$NCPU memory=$MEMORY
Preview: "D:\Programs\DYNA\program\LS-DYN~2.EXE" i=D:\TEMP\DYNA\Scracth\K\W6_BRA~1.K ncpu=4 memory=100

ID	Input File	Run Command	Status	ETA
1	D:\TEMP\DYNA\Scracth\K\W6_BracketModel	"D:\Programs\DYNA\program\LS-DYN~2.EXE" i=D:\TEMP	Finished (N	N/A

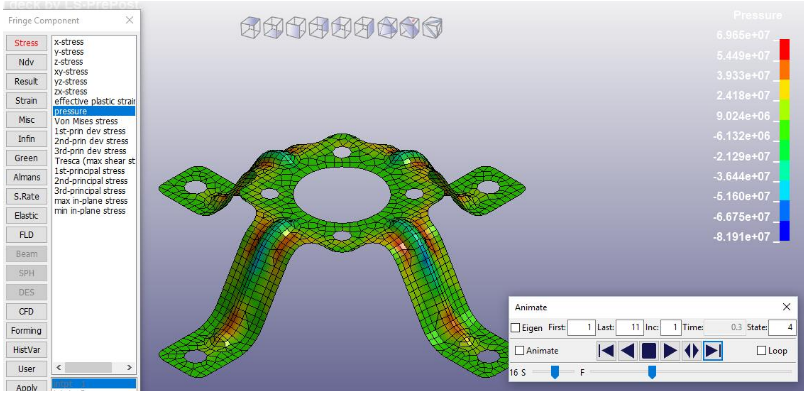
▲ Run LS-DYNA Screen

Post-Processing

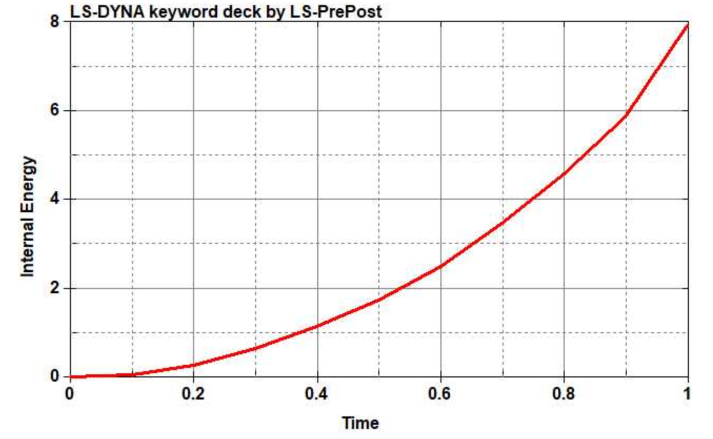
The analysis time will be relatively long compared to the size of the model. As you solve static problems with explicit, the time to find convergence may seem relatively longer than other collision analysis. When the analysis is complete, it will be displayed in green as ***Finished*** and you can check the result by selecting the **D3plot** icon directly above. When the LS-Prepost window opens and the model appears, check out the various results in the **Fringe** menu of the Post-processing option. Since the calculation interval of **D3PLOT** is set to 0.1sec, it can be seen that a total of 11 frames are generated. When 10 MPa was applied to the model, Von Mises was found out to be about 792 MPa and the maximum displacement was about 3.1 mm at the center.

- **Post → Fringe component → Pressure, von mises stress, Animate**
- **Post → Hist → Global → Internal Energy → Plot**
- **Select Post → Fringe component → Ndv → Result displacement**

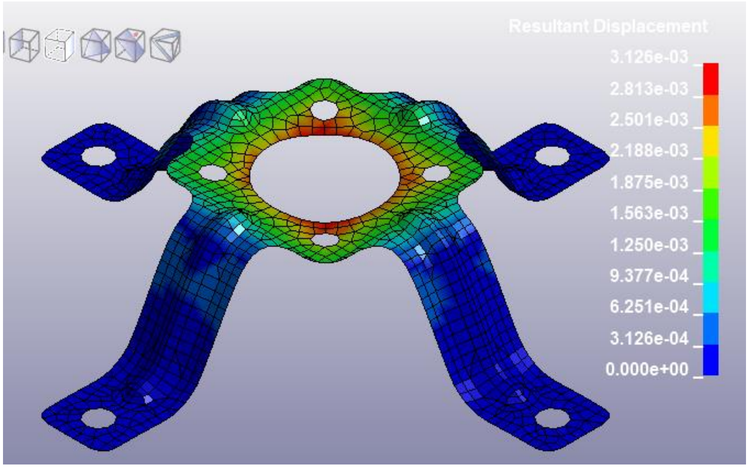
So far, we have carried out a simple load analysis on the 3D bracket. We hope that you will become familiar with the work environment by repeating more analysis as you change the shell thickness or physical properties (change the material data).



▲ Pressure distribution



▲ Strain energy distribution



▲ Displacement distribution result

Knowledge

Material Library offered by LS-DYNA

LS-DYNA provides more than 200 various material property models and also users can create property models through User-Defined Material with **FORTRAN** code if necessary. The classification is provided in the following physical properties, so learning the classification system once will help you learn. In the case of metallic materials, **MAT24_ Piecewise_Linear_Plasticity** model is frequently used for the elastoplastic material model and **MAT20_Rigid** property is also applied for the collision analysis.

Linear Elastic Models

- Isotropic (MAT1)
- Orthotropic (MAT2)
- Anisotropic (MAT2)

Nonlinear Elastic Models

- Blatz-Ko Rubber (MAT7)
- Mooney-Rivlin Rubber (MAT27)
- Viscoelastic (MAT6)

Plasticity Models

- Bilinear Isotropic (MAT3)
- Temperature Dependent Bi-Isotropic (MAT4)
- Bilinear Kinematic (MAT3)
- Plastic Kinematic (MAT3)
- Powerlaw Plasticity (MAT18)
- Rate Sensitive Powerlaw Plasticity (MAT64)
- Strain Rate Dependent Plasticity (MAT19)
- Piecewise Linear Plasticity (MAT24)

Foam Models

- Low Density Foam (MAT57)
- Viscous Foam (MAT62)
- Mooney-Rivlin Rubber (MAT27)
- Viscoelastic (MAT6)

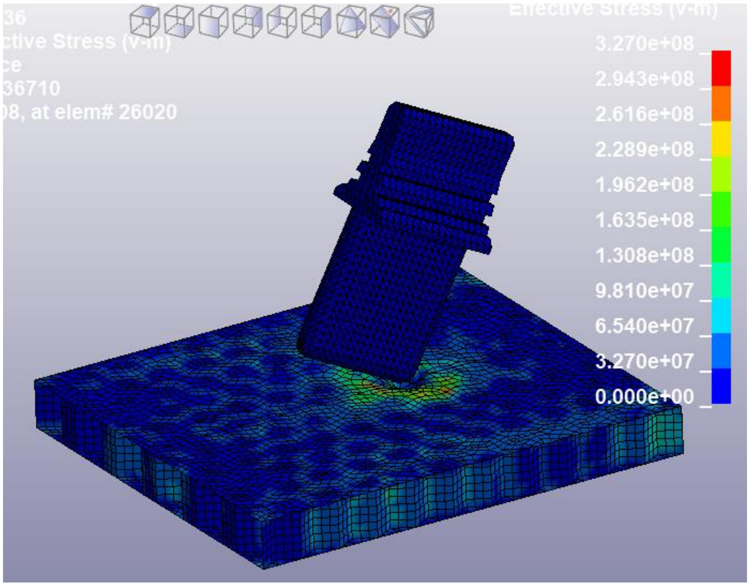
Spring Damper Models

- Linear Elastic Spring (MAT18)
- Linear Viscous Damper
- Nonlinear Elastic Spring
- Nonlinear Viscous Damper
- Elasto-plastic spring
- General Nonlinear Spring

Others

- Rigid (MAT20)
- Cable (MAT71)

Chapter 6. Blade Impact Analysis of Composite Sandwich



Workshop Objectives

- ✓ **Understanding the properties of composite materials**
- ✓ **Practice of inputting material properties of anisotropic materials**
- ✓ **Learn how to model sandwich structures**
- ✓ **Learn impact modeling process**

Duration: 60 min

Application: Composite, Explicit, Structural, Impact Analysis

Model Files

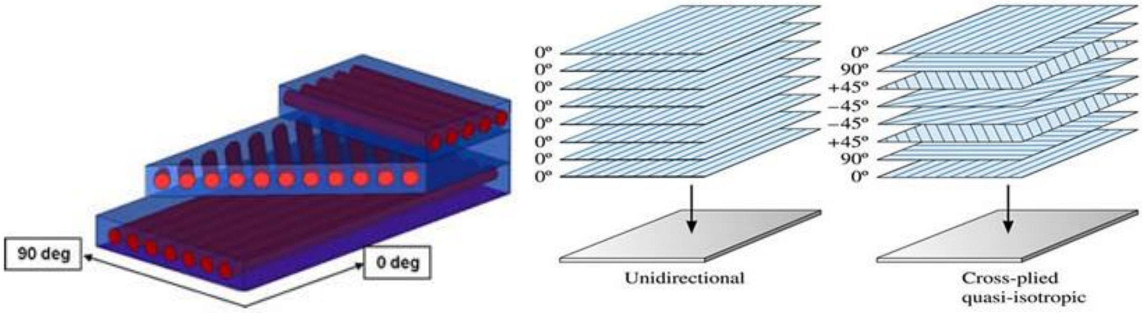
- W6_INITIAL_SandwichImpact_Mesh.k: Initial state model
- W6_SETUP_SandwichImpact.k: Final state model

Composite Modeling

Composite materials are now being more commonly used in our everyday lives so that they are no longer classified as new materials. Composite material refers to a material in which two or more materials are mixed with each other. In recent years, carbon composite materials have been used in almost all industries such as aviation, aerospace, automobile, sports, and medical care due to their lightweight, high rigidity, and high strength.

▲ Carbon composite chassis[6]

The direction of reinforcing fibers is one of the most important factors in modeling composites. LS-DYNA provides many functions for modeling composite materials.



▲ Composite material orientations

In this chapter, we will learn how to model the properties of composite structures. The impact analysis will be performed on composite sandwich structures that are mounted inside the aircraft's engine as a blade containment system. Basically, the layer modeling method provides the following three methods.

- **Layered thin shell elements**: The structure is stacked in one shell element and the simple structure reduces the analysis time, but the stress calculation in the thickness direction is not possible and the phenomenon such as the separation between layers is not simulated.

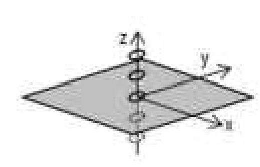
---

[6] Image courtesy of carbodydesign.com

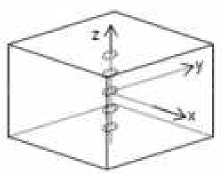
Modeled is done with the help of keyword * **Part_Composite_TShell.** Thickness, direction and various properties can be applied for each layer.

- **Layered thick shell elements:** The structure or 3D stress that is stacked in one shell element is calculated and the stress can be calculated for each stack. It is modeled as ***Part_Composite** and it is possible to apply thickness, direction and various properties by layer. No separate section property input is required. Also, it is also possible to model various stacking patterns in one part using the ***SECTION_Shell_composite** keyword.

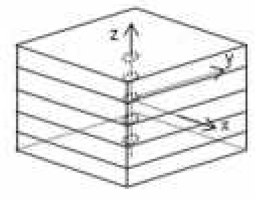
- **Solid elements:** It is a structure that creates elements for each layer, so that phenomenon simulation such as delamination is possible, but the analysis time is relatively long.



▲ Layered Thin Shell



▲ Layered Thick Shell



▲ Solid Elements

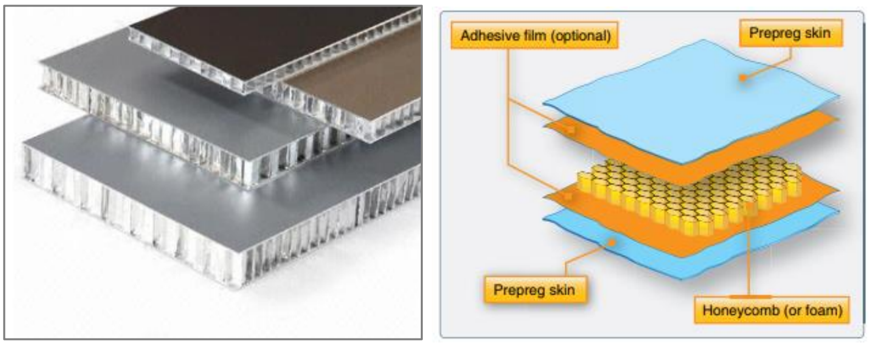
W6

Workshop 6.

Blade Impact Analysis of Composite Sandwich

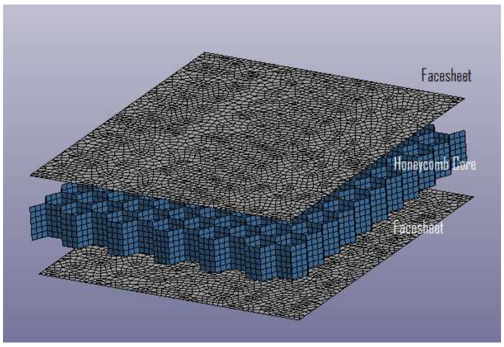
Introduction

In this chapter, the face sheet of the sandwich structure is modeled using layered thick shell element modeling and the internal honeycomb core is modeled using the general shell element. The purpose of the analysis is to evaluate the damage pattern when a blade with a speed of 18 m/s hits the honeycomb sandwich panel.



▲ Honeycomb Sandwich panel

Sandwich structure is widely used in almost all industries such as aviation/automobile/construction/shipbuilding due to its excellent bending and flexural properties against weight. In this tutorial, the user will learn how to model composite materials, input properties, and analyze impact absorption model for honeycomb sandwich structures.

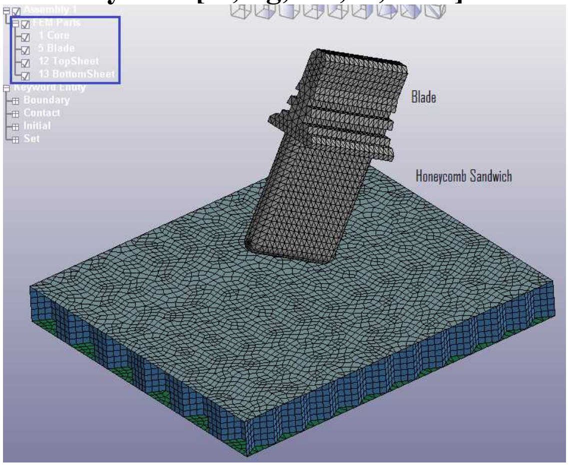


▲ Honeycomb Sandwich model

Import Mesh Model

Let's import the ***W6_INITIAL_SandwichImpact_Mesh.k*** model, which already has a mesh created. A model consisting of four parts with meshes appears as shown below. It is a model where parts are located to simulate the situation just before the impact moment. The SI model unit is used for the analysis model.

- **Unit system [m, kg, sec, N, GPa]**



▲ Honeycomb Sandwich and blade mesh model

Material Property

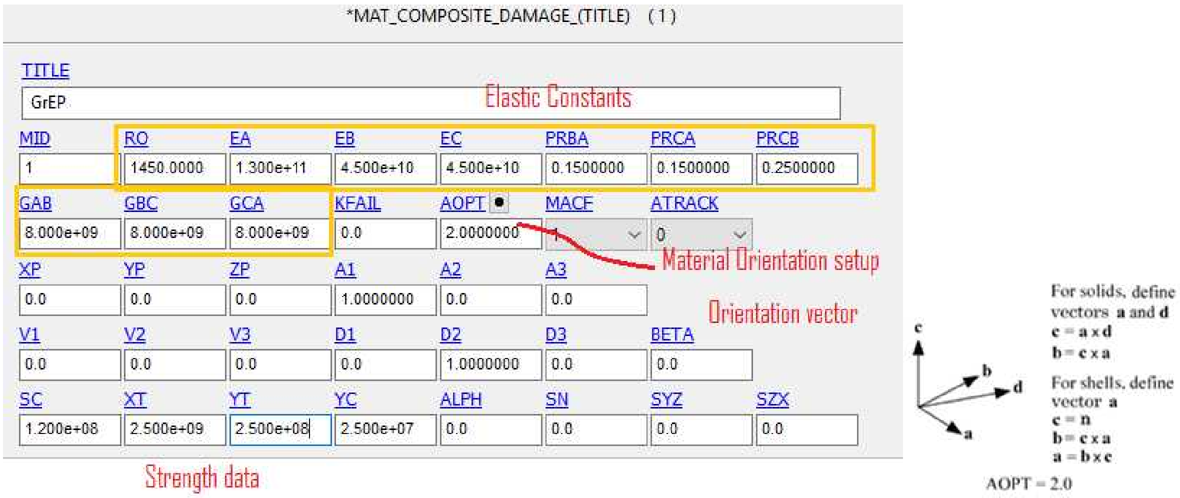
Composite physical properties can be modeled in various ways. Fundamentally, composite materials, unlike metals, have orthotropic properties, not isotropic. There are many physical property models that can define such anisotropy. In this exercise, the material model is created using **MAT022_ Composite_Damage** property.

- **Keyword Manager → MAT | 022_Composite_Damage → Input property**

*Axial Modulus: EA, EB, EC, PRBA, PRCA, PRCB, GAB, GBC, GCA

*Orientation configuration options: AOPT = 2, various options, where AOPT = 2 defines the principal axis a vector and the orthogonal d vector on lines 3-4.

*In line 5, input the strength value in the axial direction.



*MAT_COMPOSITE_DAMAGE_(TITLE) (1)

TITLE

GrEP

MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
1	1450.0000	1.300e+11	4.500e+10	4.500e+10	0.1500000	0.1500000	0.2500000
GAB	GBC	GCA	KFAIL	AOPT ●	MACF	ATRACK	
8.000e+09	8.000e+09	8.000e+09	0.0	2.0000000	1	0	
XP	YP	ZP	A1	A2	A3		
0.0	0.0	0.0	1.0000000	0.0	0.0		
V1	V2	V3	D1	D2	D3	BETA	
0.0	0.0	0.0	0.0	1.0000000	0.0	0.0	
SC	XT	YT	YC	ALPH	SN	SYZ	SZX
1.200e+08	2.500e+09	2.500e+08	2.500e+07	0.0	0.0	0.0	0.0

Elastic Constants

Material Orientation setup

Orientation vector

Strength data

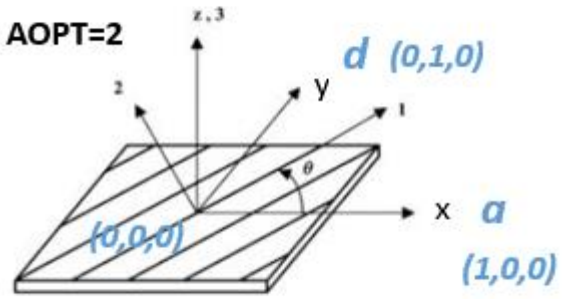
For solids, define vectors a and d
c = a x d
b = c x a

For shells, define vector a
c = n
b = c x a
a = b x c

AOPT = 2.0

▲ MAT022_material property

AOPT = 2 was used to specify the direction of reinforcing fibers. It is very important to set the local coordinate because the fiber direction of the composite material is a key factor in determining the physical properties. If **AOPT** = 2 is selected, the principal axis (a) and the d vector perpendicular to the a-axis is defined as shown in the figure below and the lamination angle is entered using the angle created with the main axis.



▲ AOPT = 2 option coordinates for fiber orientation reference

The variables that define the axial strength of the composite are as follows:

- SC: Shear strength
- XT: Axial tensile strength
- YC: Transverse compressive strength
- YT: Transverse tensile strength

The **MAT22** model uses Chang-Chang's Failure model. The axial tension/compression failure and matrix tension/compression failure of the reinforcing fibers are defined by the following criteria:

- **Tensile fiber mode:**

$$\sigma_{aa} > 0 \ \ then \ \ e_f^2 = \left(\frac{\sigma_{aa}}{X_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases}$$

On failure: $E_1 = E_2 = G_{12} = \nu_{12} = \nu_{21} = 0$

- **Compressive fiber mode:**

$$\sigma_{aa} < 0 \ \ then \ \ e_c^2 = \left(\frac{\sigma_{aa}}{X_{tc}}\right)^2 - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases}$$

On failure: $E_1 = \nu_{12} = \nu_{21} = 0$

- **Tensile matrix mode:**

$$\sigma_{bb} > 0 \ \ then \ \ e_m^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases}$$

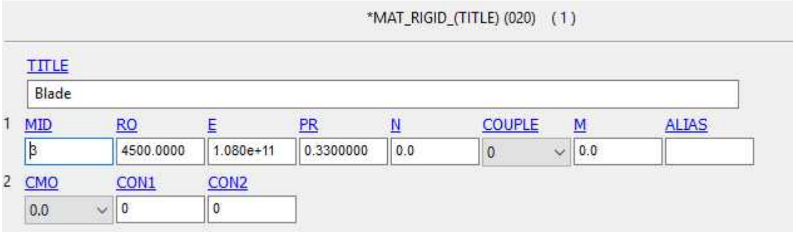
- **Compressive matrix mode:**

$$\sigma_{bb} < 0 \ \ then \ \ e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \ failed \\ < 0 \ elastic \end{cases}$$

On failure: $E_2 = \nu_{21} = \nu_{12} = 0 = G_{12} = 0$

The blade is defined as a rigid body having the properties of titanium. The honeycomb structure of the sandwich core is modeled with **MAT024** consisting of aluminum properties.

- **Keyword Manager → MAT | 020-Rigid → input material property**
- **Keyword Manager → MAT | 024-Piecewise_Linear_Plasticity →input material property**

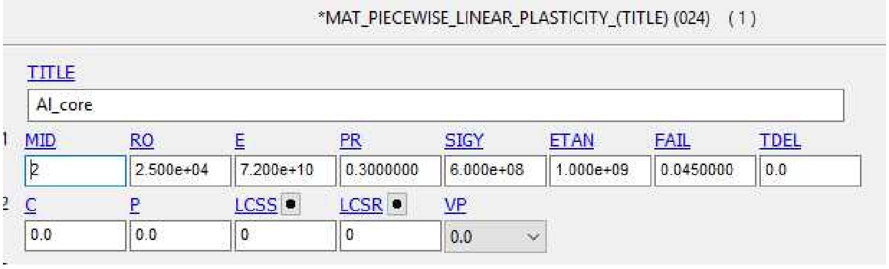


*MAT_RIGID_(TITLE) (020) (1)

TITLE: Blade

1	MID	RO	E	PR	N	COUPLE	M	ALIAS
	3	4500.0000	1.080e+11	0.3300000	0.0	0	0.0	
2	CMO	CON1	CON2					
	0.0	0	0					

▲ MAT020_ Material Property



*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE: Al_core

1	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
	2	2.500e+04	7.200e+10	0.3000000	6.000e+08	1.000e+09	0.0450000	0.0
2	C	P	LCSS	LCSR	VP			
	0.0	0.0	0	0	0.0			

▲ MAT024_Material Property

NOTE Material Properties for Orthotropic Material in LS-DYNA

LS-DYNA offers various composite material properties.

• **MAT002** (Elastic Orthotropic) The axial properties can be input individually to the basic elastic model, but damage or breakage cannot be modeled. Nine axial properties are entered as triaxial modulus: EA, EB, EC, axial Poisson's ratio: PRBA, PRCA, PRCB, axial Shear Modulus: GAB, GBC, GCA.

The following models can be variously modeled for damage and are classified according to the damage modeling method implemented:

- **MAT022_Composite_Damage**
- **MAT054, 055_Enhanced_Composite_Damage**
- **MAT058_Laminated_Composite_Fabric**
- **MAT158_Laminated_Composite_Fabric including strain rate effects**
- **MAT059_Composite_Failure (Shell, Solid) _Model**

Among these, **MAT22, 54, 55** are properties applicable to all shells, tshells, and solids. In particular, **MAT54 / 55** properties can be effectively analyzed by applying relatively simple input parameters and damage models to complex composite failure mechanisms. Therefore, the **MAT54 / 55** models have been widely applied to the collision analysis of composite materials in the industrial field. Compared to **MAT22**, the other models have more damage mechanisms that can be expressed.

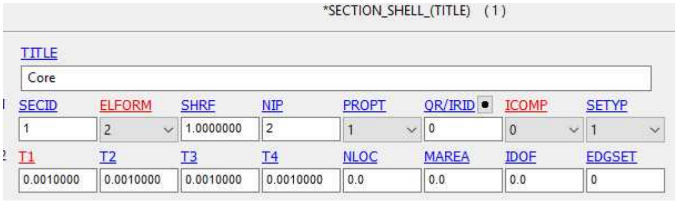
In addition, there is * **MAT116**, which is the concept that the entire lamination is calculated and analyzed through the lamination theory and here individual layer stresses are not calculated, but the advantage is that analysis time can be considerably shortened. This would be useful for very thick stacking materials.

As DYNA is upgraded, the latest composite models continue to be added. In the case of the recently included composite model, * **MAT_261_Laminated_Fracture_Daimler_Pinho / Camanho**, it is a physical model that can model not only interlaminar failure but also intralaminar failure damage. As the name suggests, this is a property model developed by Mercedes Benz, Germany, to analyze composite materials. This model is also applicable to Shell, tshell and Solid.

Section Property

The top panel and bottom panel of the sandwich panel consist of laminated composite materials and the middle honeycomb core consists of 1mm aluminum cell. Part composed of laminated composite material does not need to input separate section property because of all the stacked information required in * **Part_Composite**. For the Impactor blade, enter **Solid Section** and enter **ELFORM** = 16 because the mesh was created as a Tetrahedron mesh.

- **Keyword Manager → Section → Shell → ELFORM=2, T1=0.001 → Accept**
- **Keyword Manager → Section → Solid → ELFORM=16 → Accept**



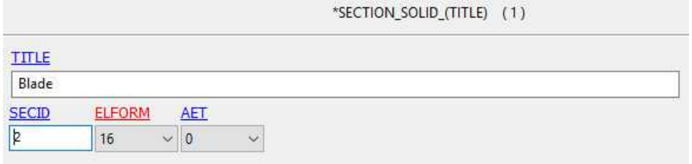
*SECTION_SHELL_(TITLE) (1)

TITLE: Core

SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
1	2	1.0000000	2	1	0	0	1

T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
0.0010000	0.0010000	0.0010000	0.0010000	0.0	0.0	0.0	0

▲ Section_Shell for Honeycomb core



*SECTION_SOLID_(TITLE) (1)

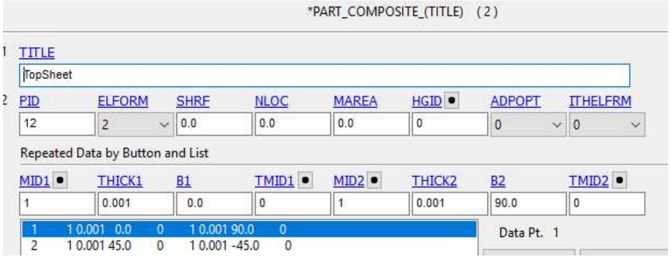
TITLE: Blade

SECID	ELFORM	AET
2	16	0

▲ Section_Solid for Blade

Now that the properties and section properties have been created, it is a stage to assign the properties that correspond to each part. The first step is to enter the * **Part_Composite** keyword, which defines the properties of the sandwich panel. In **Part_Composite**, you can input section characteristics such as ELFORM of the part and input the number of materials, thickness, fiber direction/orientation, etc. for each stack. The direction of lamination is from the bottom part based on the normal vector as shown below:

- **Keyword Manager → Part_Composite → ELFORM=2, MAT=1, Thick1=0.001, B1= 0 ….. → input values(see image), create Part_Composite repeatedly**



*PART_COMPOSITE_(TITLE) (2)

TITLE: TopSheet

PID	ELFORM	SHRF	NLOC	MAREA	HGID ●	ADPOPT	ITHELFRM
12	2	0.0	0.0	0.0	0	0	0

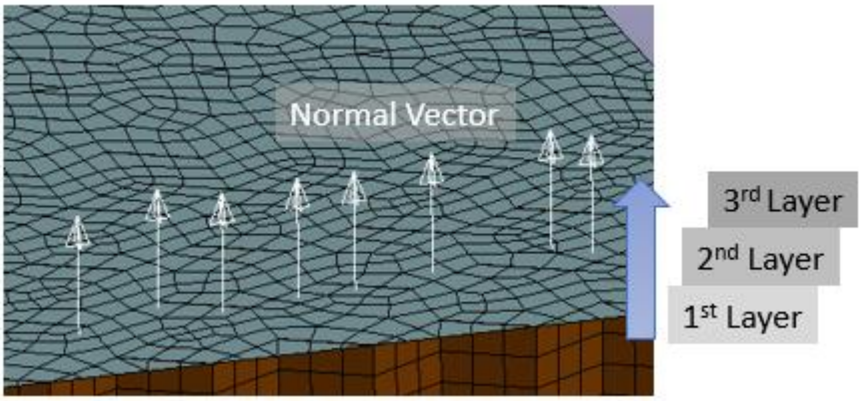
Repeated Data by Button and List

MID1 ●	THICK1	B1	TMID1 ●	MID2 ●	THICK2	B2	TMID2 ●
1	0.001	0.0	0	1	0.001	90.0	0

```
1    1 0.001  0.0    0    1 0.001 90.0    0
2    1 0.001 45.0    0    1 0.001 -45.0   0
```

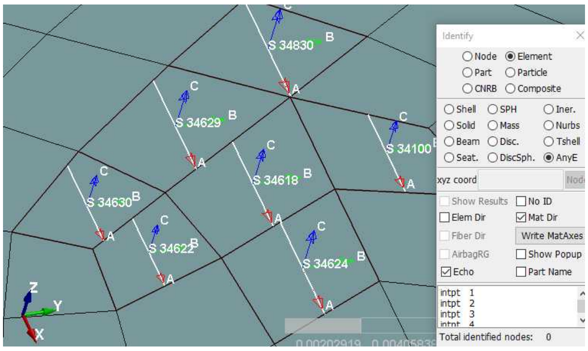
Data Pt. 1

▲ Part_Composite Keyword Input Screen (Lamination Angle [0/90 / + 45 / -45])



▲ Part_Composite layer order

The fiber direction can be found by **Element tools → Identify → Element | Mat Direction.** The set direction is displayed as shown below. If Int1 ~ Int4 is selected in the dialog box, the axial direction set for each layer appears. Check if it matches: **[0 / 90 / -45 / + 45].**



○ Node ● Element
○ Part ○ Particle
○ CNRB ○ Composite
○ Shell ○ SPH ○ Iner.
○ Solid ○ Mass ○ Nurbs
○ Beam ○ Disc. ○ Tshell
○ Seat. ○ DiscSph. ● AnyE
xyz coord
☐ Show Results ☐ No ID
☐ Elem Dir ☑ Mat Dir
☐ Fiber Dir Write MatAxes
☐ AirbagRG ☐ Show Popup
☑ Echo ☐ Part Name

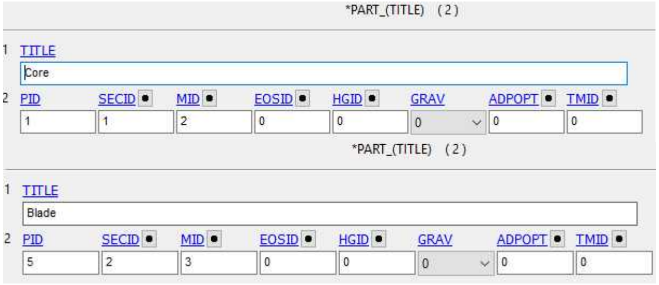
```
intpt  1
intpt  2
intpt  3
intpt  4
```

Total identified nodes: 0

▲ Checking Material direction

Next, select and enter the section characteristics of the honeycomb core and for the blade.

- **Keyword Manager → Part → select SECID and MID corresponding to Core and Blade**



*PART_(TITLE) (2)

1 TITLE: Core

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
1	1	2	0	0	0	0	0

*PART_(TITLE) (2)

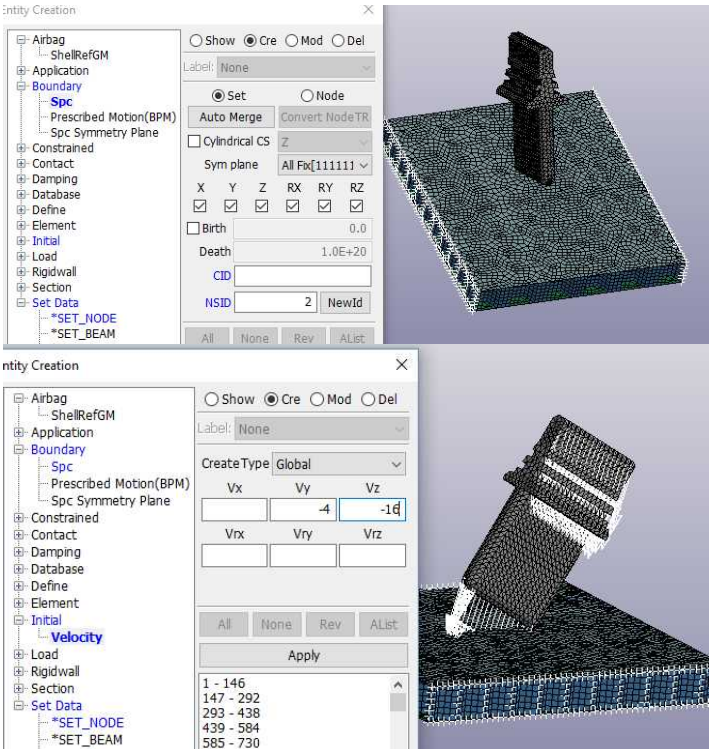
1 TITLE: Blade

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
5	2	3	0	0	0	0	0

Boundary Condition

Fix both side ends of the sandwich panel and enter the blade impact speed.

- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Area → select both side nodes(see image), 6 D.O.F all check → Apply**
- **Model and Part → Create Entity → Initial | Velocity → Sel. Part | Pick → select blade, Vy=-4, Vz=-16 → Apply**

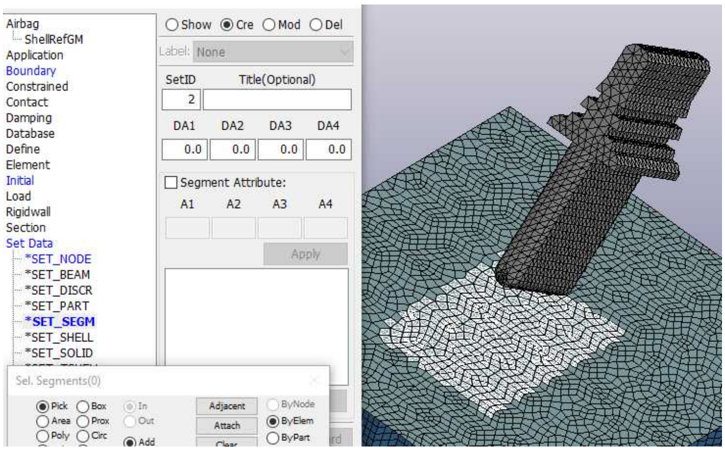


▲ Sandwich panel boundary condition and blade initial velocity input

Contact Condition

In this model, the blade colliding with the sandwich panel consists of three parts. Therefore, it is effective to set the contact condition of the **Segment-Part** by setting the area of the panel where the collision occurs to **Segment option**. Set the segment where the collision is expected to occur as shown in the figure below.

- **Model and Part → Create Entity → Set Data → Segment → Sel. Segments | Area | Add | By Element → select impact zone(see image) → done**



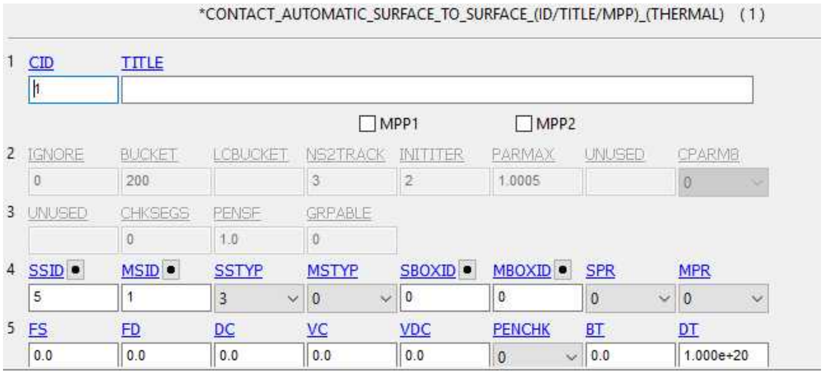
SetID 2, Title(Optional)

DA1	DA2	DA3	DA4
0.0	0.0	0.0	0.0

▲ Segment setting of sandwich panel

Contact condition is created by using * **Automatic_Surface_to_Surface**, select **MSTYP** = 0 (segment) and **SSTYP** = 3 (Part) and assign the created segment and blade part respectively.

- **Keyword Manager → Contact → Automatic_Surface_to_Surface → Input values (see image)**


*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (1)

1	CID	TITLE						
	1							
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	5	1	3	0	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.0	0.0	0.0	0.0	0.0	0	0.0	1.000e+20

Analysis Condition

Since the analysis time is 0.0021sec, enter **ENDTIM** = 0.0021 in the * **Control_termination** keyword and set **d3plot**, **ASCII data** and **Extent_Binary** in the result file database.

- **Keyword Manager → Control | Termination → ENDTIM = 0.002**
- **Keyword Manager → ASCII_option → Default DT=1e-4, GLSTAT, MATSUM, RCFORC**
- **Keyword Manager → BINARY_D3PLOT → NPLTC=50 (Generate 50 frame)**
- **Keyword Manager → Extent_Binary → NEIPS=2, CMPFLG=1**

* NEIPS=2 represents the number of integration points for the history output of the damage factor in the composite (T) shell. In the case of solid element, it is defined as NEIPH and CMPFLG = 1 means the stress and displacement calculated based on local coordinate. If CMPFLG = 0 is selected, the output is based on global coordinates.

- **Keyword Manager → KEYWORD → MEMORY=800000000, NCPU=4 (Enter to fit user PC)**

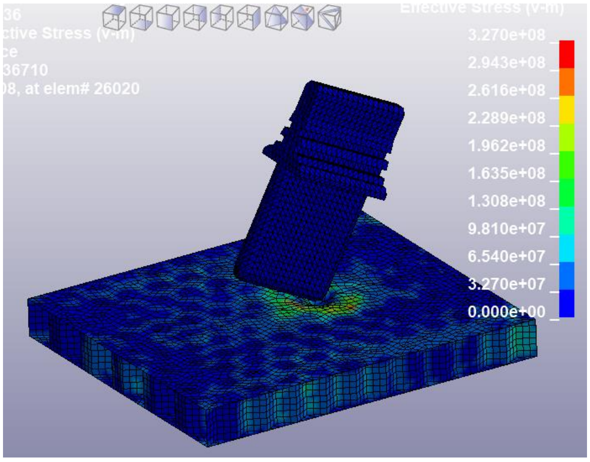
So far, we have practiced keyword modeling for sandwich panel and blade impact analysis, including composite materials modeling. Then, save the completed **Keyword** and perform the interpretation. Let's open the generated keyword and the provided '***W6_SETUP_SandwichImpact.k***' model with a text editor to compare with each other and see if there are any differences.

Analysis Result

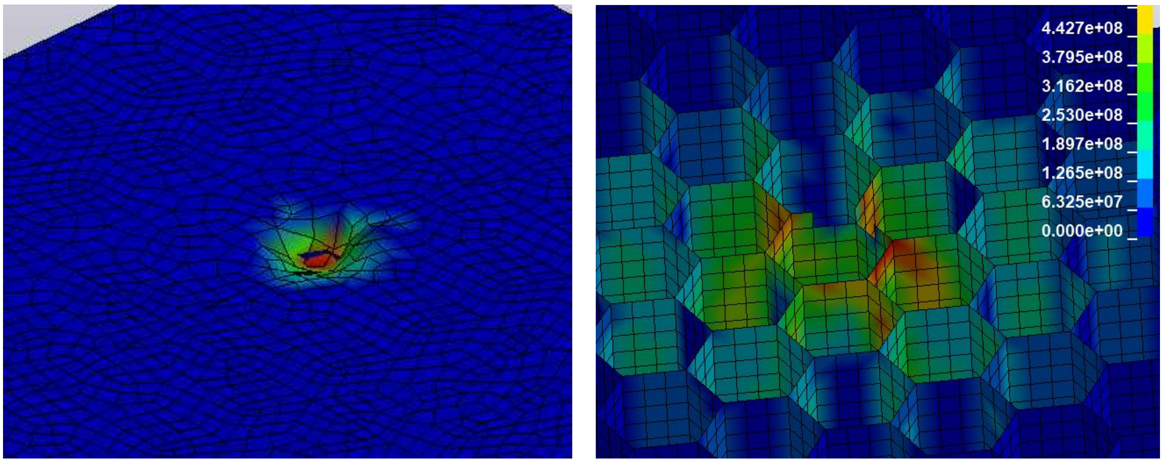
After the analysis is completed, let's load **D3plot** into Prepost. You can also load it directly by clicking the **LSPP** icon in the LS-Run dialog. Let's select Pressure, Von mises stress, etc. in the Fringe component of the post and show the calculated 50 frame result.

- **Post → Fringe component → Pressure, von mises stress→ select Animate**

As shown in the figure below, you can see that the blade bounces after a certain point of time after it damages the sandwich panel upon impact. Since damage intensity value is entered during composite panel material modeling, it can be seen that the element disappears as the panel is damaged when a certain level of stress occurs. Similarly, since the honeycomb core cell has the failure strain value input, which is the damage criterion, the core cell part where the element is deleted can be identified as shown below:

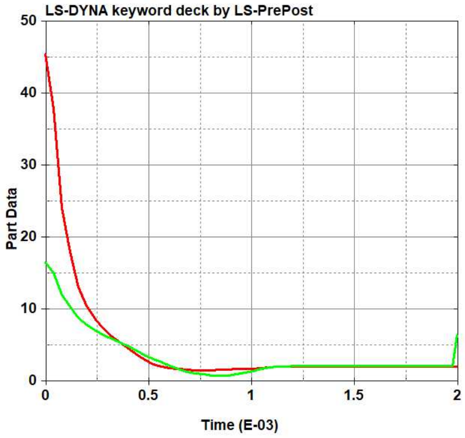


Stress distribution



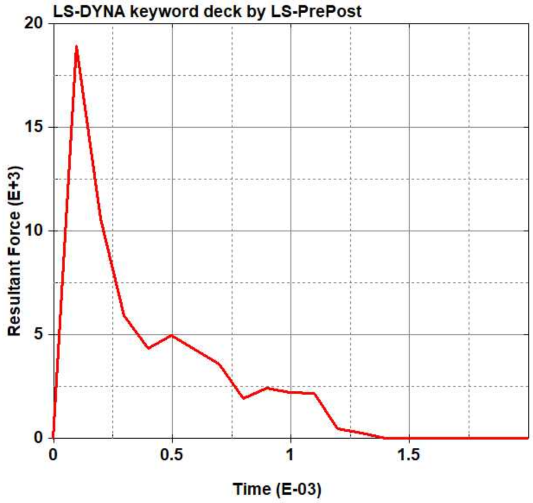
Damage in the composite panel and the honeycomb core

Check the Part in the History output and select the Blade to plot the Kinetic energy and Rigid body velocity. You can see how energy absorbs and slows down as collisions occur.



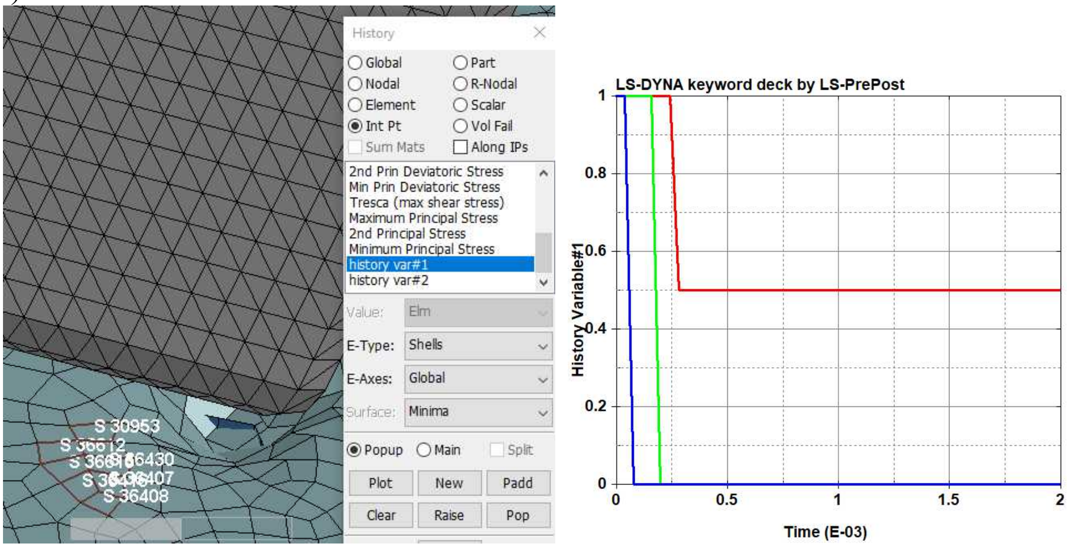
Change of kinetic energy and speed of the blade

After selecting **RCFORC** in **ASCII**, load and plot the resultant force of Ma-1 (Master Surface). The impact load on the sandwich target panel can be checked as follows.



▲ Impact load distribution in the sandwich panel

To identify the damage mechanism of the composite panel, **NEIPS** = 2 was entered in the * **Database_Extent** keyword, which identifies the classification of **Chang-Chang's failure mechanism** as a history output. Go to **History output** → **Int. Pt** → **history var** # 1 (tensile fiber mode), # 2 (compressive fiber mode), # 3 (tensile matrix mode), # 4 (compressive matrix mode) can be checked. If you select the elements near the damaged part as shown in the figure below, you can check by which mechanism and at what point (1: Elastic → 0: failed).

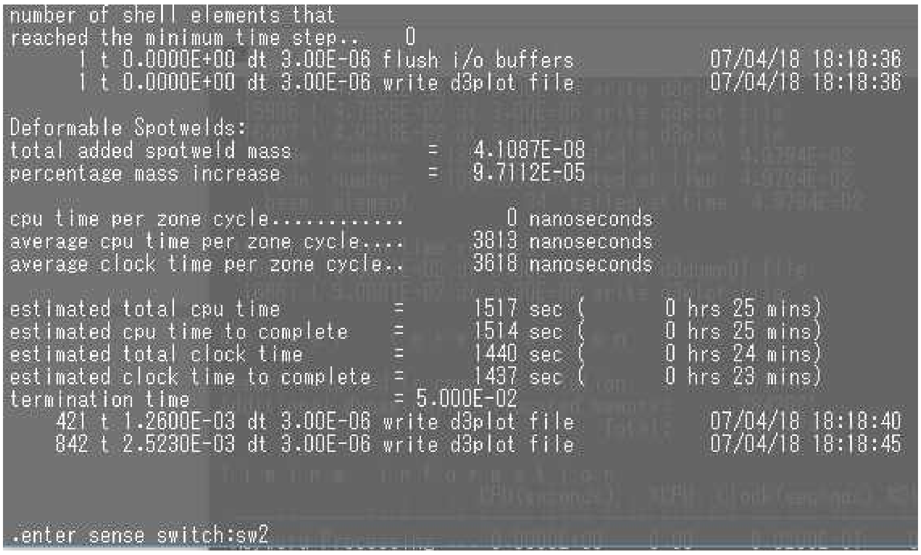


▲ MAT22_Chang-Chang 's damage mechanism count check (when NIEPS = 2)

In this workshop tutorial, the impact load analysis of the blades for the composite sandwich structure was practiced. It is hoped that the practice has improved understanding of the modeling and properties input procedures for composite structures.

NOTE Controlling the Analysis Process

LS-DYNA analysis sometimes takes quite a long time to solve. Therefore, it is necessary to take action during the analysis process to make sure that the analysis model is working as intended. In general, if multiple D3PLOTs are generated in the following Log screen, the analyst must check this. If it is determined that there is a problem in the analysis process, the analysis should be stopped.



```
number of shell elements that
reached the minimum time step..    0
      1 t 0.0000E+00 dt 3.00E-06 flush i/o buffers            07/04/18 18:18:36
      1 t 0.0000E+00 dt 3.00E-06 write d3plot file            07/04/18 18:18:36

Deformable Spotwelds:
total added spotweld mass          =   4.1087E-08
percentage mass increase           =   9.7112E-05

cpu time per zone cycle............          0 nanoseconds
average cpu time per zone cycle....       3813 nanoseconds
average clock time per zone cycle..       3618 nanoseconds

estimated total cpu time          =      1517 sec (       0 hrs 25 mins)
estimated cpu time to complete    =      1514 sec (       0 hrs 25 mins)
estimated total clock time        =      1440 sec (       0 hrs 24 mins)
estimated clock time to complete  =      1437 sec (       0 hrs 23 mins)
termination time                  = 5.000E-02
    421 t 1.2600E-03 dt 3.00E-06 write d3plot file            07/04/18 18:18:40
    842 t 2.5230E-03 dt 3.00E-06 write d3plot file            07/04/18 18:18:45


.enter sense switch:sw2
```

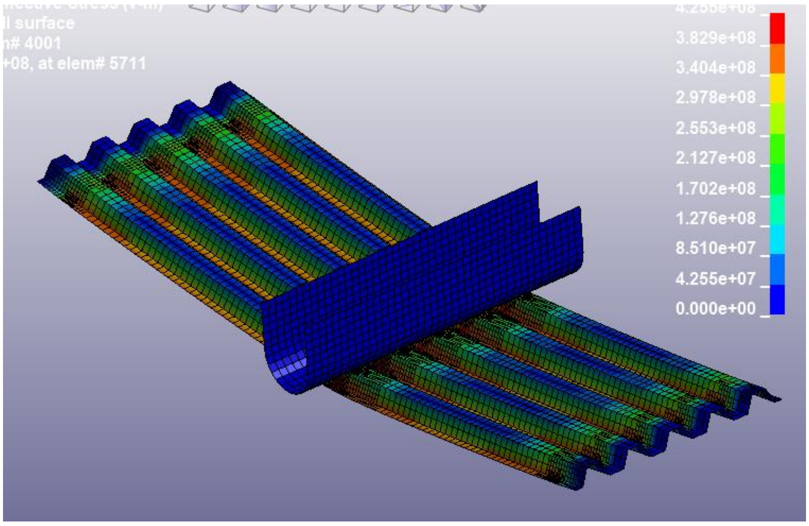
During the analysis, in the process window, press **Control + C** to stop the analysis and use the **sense** switch.

- **SW1. Restart Save the file and stop parsing.**
- **SW2. Display the time and cycle of the current state and continue the analysis.**
- **SW3. Save the Restart file and continue the analysis.**
- **SW4. Save the plot state and continue the analysis.**
- **Stop. Pause analysis and close open files.**
- **Quit. Analysis stops completely.**

In general, the initial analysis time displayed in the log screen is calculated to be very large, so it may be helpful to check the expected analysis time of the current state by entering sw2 after Control + C.

If you stop the analysis through Stop or Quit command, the result file in that state will remain in the folder so you can check it later for debugging.

Chapter 7. 3P Bending Analysis (Explicit vs. Implicit)



Workshop Objectives

- ✓ **Understand the concepts of explicit and implicit.**
- ✓ **Practice how to create 3P Bending Test**
- ✓ **Applying Adaptive Meshing Technique**
- ✓ **Understand the difference between Explicit and Implicit models.**

Duration: 90min

Application: Structure, Explicit, Implicit, 3P Bend Test

Model Files

- W7_INITIAL_MeshPart.k : Initial state (Mesh Model)
- W7_SETUP_3PBendTest_Explicit.k : Final state model (Explicit)
- W7_SETUP_3PBendTest_Implicit.k : Final state model (Implicit)

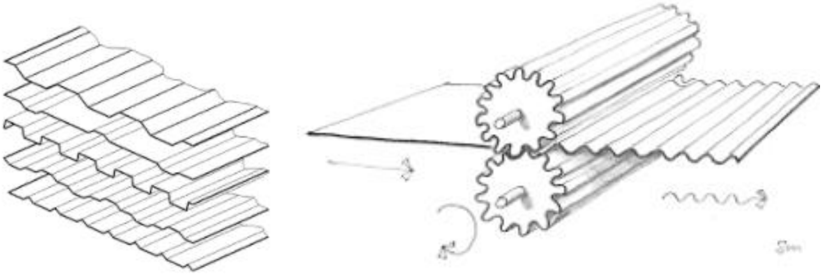
W7

Workshop 7.

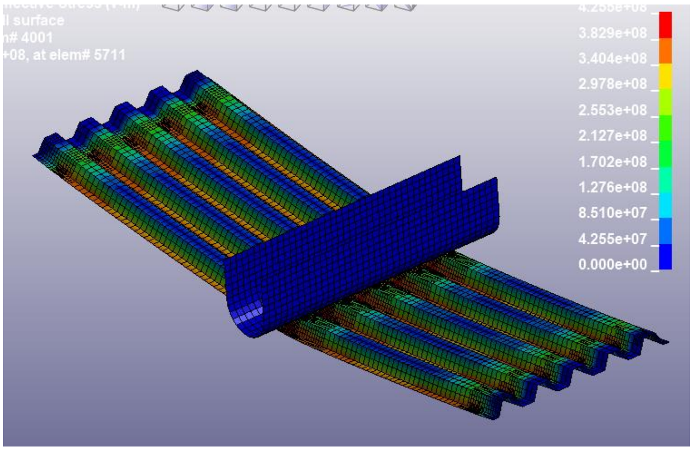
3 Point Bending Analysis with Explicit & Implicit

Introduction

In the engineering field, tension and bending tests of many mechanical components (test specimens) are often done to find out its behavior and characteristics. In this tutorial, we will practice the analysis of the three-point bending test on corrugated panels. The corrugated panel is one of the representative structural forms widely used in the industrial and architectural fields. Here, the panel is bent in various forms to increase the rigidity of the bending direction. We will practice the analysis of the three points bending test by applying both Implicit and Explicit techniques. Then, we shall examine the differences between implicit and explicit methods. Also, we will practice how to incorporate the Adaptive remeshing technique, which is one of the effective techniques in LS-DYNA for getting good results.



▲ Various corrugated panels and processing methods



▲ Corrugated panel 3-point bending test analysis model

In this analysis model, the ***MAT24_Piecewise Linear Plasticity** model, which is the representative plastic property model provided by LS-DYNA, is applied to the explicit and implicit solver. Then, we will learn how to extract the graph for deformation and bending load from the solved analysis results.

A. Explicit Solver Model

Import Geometry Model

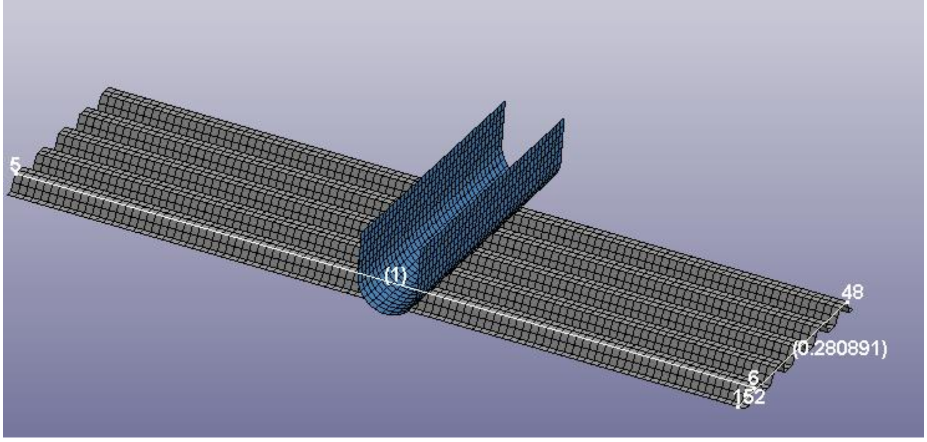
The model consists of two parts, a corrugated panel and an indenter that loads at the center, as shown below. The model can be verified by importing the provided file ***'W7_Initial_mesh.k'***. The unit used in the model is [mm, kg, ms] unit system.

• Unit system [mm, kg, ms, kN, GPa, kN-mm]

To check the size of the whole part, let's measure the model using the **Measure** function. The measured size is 1m in the length direction of the panel and about 0.3m in the width direction.

. ◀Measure icon

- **Element tools → Measure | Dist N2N | Measure the distance between nodes by clicking on both ends of the panel**

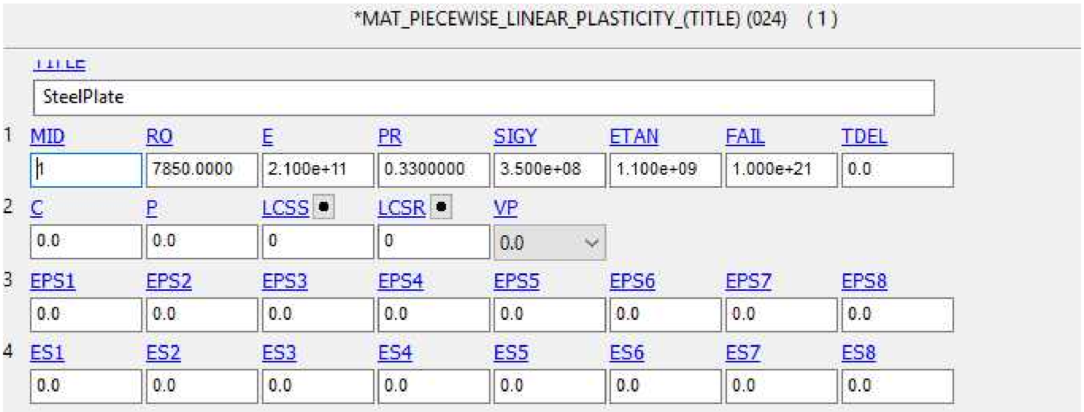


▲ Imported 3-point bending test analysis model

Material Property

Let's assume that the corrugated panel is made of stainless steel and input the properties using **MAT24** material card. At this time, pay attention to the unit system and its consistency. The definition of the plastic zone of **MAT24** material can be applied in various ways. The first method is to enter it using **ETAN** (Tangent Modulus) on the following input screen. The second method is to define and input **LCSS**, that is, the curve of the plastic zone and the third is to input **EPS1-ES1** (Plastic strain-corresponding yield stress) individually. In this exercise, the method of inputting **ETAN** is applied. For the properties of the indenter enter the rigid body properties.

- **Keyword Manager → MAT | 024-Piecewise_linear_plasticity → input properties**



*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024)   (1)

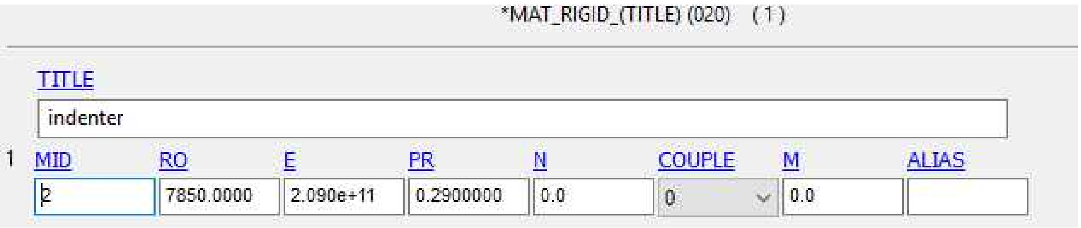
TITLE

SteelPlate

1	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
	1	7850.0000	2.100e+11	0.3300000	3.500e+08	1.100e+09	1.000e+21	0.0
2	C	P	LCSS ●	LCSR ●	VP			
	0.0	0.0	0	0	0.0			
3	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
4	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

▲ MAT024 Material Properties

- **Keyword Manager → MAT | 020-Rigid → input properties**



*MAT_RIGID_(TITLE) (020)   (1)

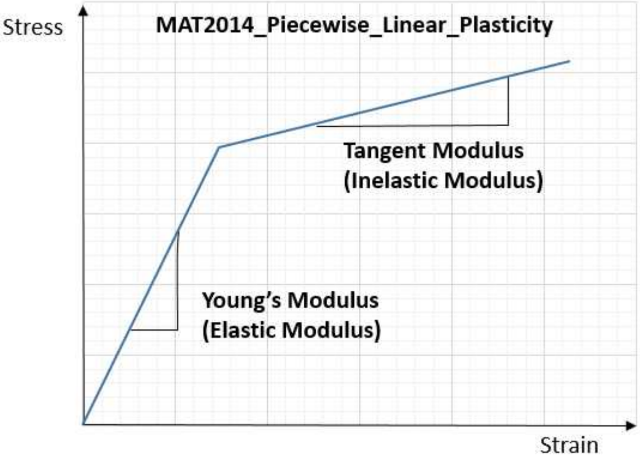
TITLE

indenter

1	MID	RO	E	PR	N	COUPLE	M	ALIAS
	2	7850.0000	2.090e+11	0.2900000	0.0	0	0.0	

▲ MAT020 Material properties

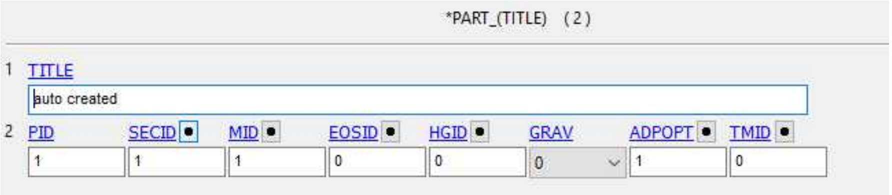
NOTE Enter the property values paying attention to the unit system. The density of steel is 7850kg / m^3, the modulus of elasticity is 210GPa, the yield strength SIGY = 350MPa and the ETAN (Tangent modulus) is the modulus 1.1GPa of the plastic zone. Tangent Modulus means Linear Plasticity. It means that the plastic area is expressed by one slope. In fact, many engineers apply the various physical models without knowing the physical meaning of them and in fact, this lack of understanding has resulted in many cases of getting numerical errors. From the graph below, you can easily understand the physical meaning of **MAT024**, the most widely used material in Dyna.



Element Property

Since both panel and load application indenter are made of shell elements, define shell sections for these. Create two sections because they are 3mm and 5mm thick, respectively. **ELFORM** chooses the default shell element of 2 (the default value). Set the section and material created in this way. Note that **ADPOPT** = 1 must be set for the corrugated panel.

- **Keyword Manager → Section → Shell → ELFORM=2, T1=0.003 → Accept, T1=0.005**
- **Keyword Manager → Part → PID=1, SECID=1, MID=1, ADPOPT=1**
- **Keyword Manager → Part → PID=2, SECID=2, MID=2, ADPOPT=0**


*PART_(TITLE) (2)

1 TITLE

auto created

PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
1	1	1	0	0	0	1	0

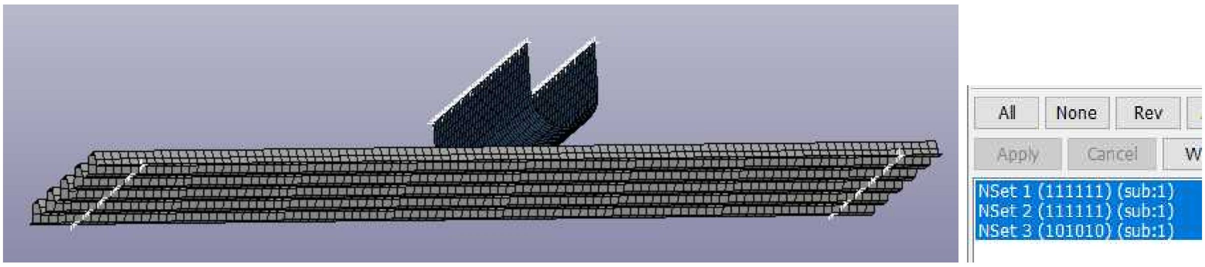
▲ Part property setting screen (Wrinkle panel, ADPOPT = 1)

ADPOPT = 1 (default = 0) is set for the component where **adaptive remeshing** occurs and the mistake of adaptive remeshing that most beginners make is also caused by not setting this parameter.

Boundary Condition

Fix all nodes on a specific line located on the left and right sides of the panel as shown below. In the case of test specimens fixed with rollers, the corresponding direction shall be constrained. In the same way, select the upper edge line of the indenter and select DOF as shown below.

- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Area → Select the node of the 5th line on the left and right of the panel as shown below, check all 6 D.O.F → Apply**
- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Area → Select the top line of the indenter as shown below, check D.O.F = 1 0 1 0 1 0 → Apply**

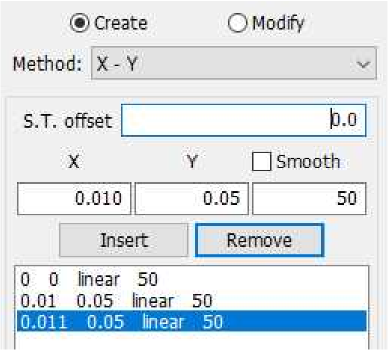


NSet 1 (111111) (sub:1)
NSet 2 (111111) (sub:1)
NSet 3 (101010) (sub:1)

▲ Boundary condition setting

The movement of the indenter is defined by creating a curve. Use **CurveGen** function of the application to create a curve(similar to ***Define_curve**).

- **Application → Tools → CurveGen | Method= X-Y | Uncheck Smooth → 0, 0 insert, 0.01, 0.05 insert, 0.011, 0.05 insert → done**



● Create ○ Modify
Method: X - Y
S.T. offset 0.0
X Y ☐ Smooth
0.010 0.05 50

0 0 linear 50
0.01 0.05 linear 50
0.011 0.05 linear 50

▲ CurveGen generation

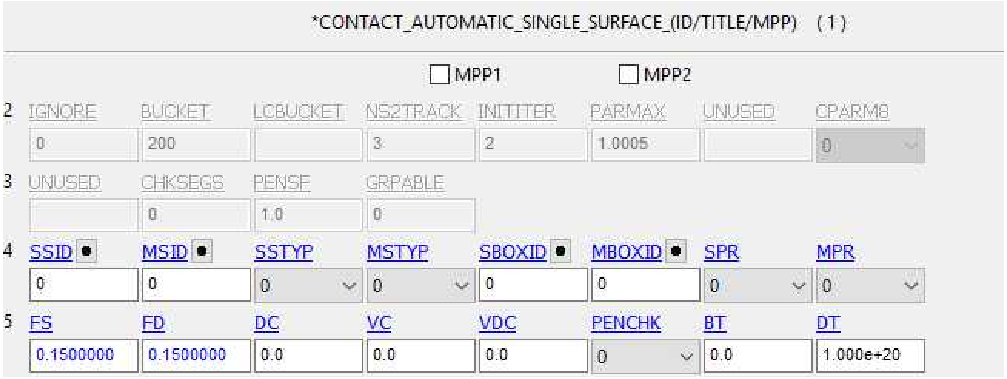
Curves can be created from **Define → Curve**, but learn how to use them because **CurveGen** function in prepost provides various options. If you check the curve created in **Define → Curve**, you can see that 50 points are automatically generated per zone. Let's set the displacement to move the indenter 10 cm per 0.01 sec using the generated curve. In the actual tensile or bending test, the moving speed of the load cell head is generally slow, which is applied at 5 ~ 10mm / min, but it is assumed to be 0.01sec in the analysis. This is not an analysis considering the strain effect, but it does not mean much by considering the strain rate effect. However, in the case of physical properties affected by the strain rate, the actual time should be set.

- **Model and Part → Create Entity → Boundary | Prescribed_Motion_Rigid | PID=2, DOF=2 (y-axis), VAD=2(displacement), LCID=1(generated Curve), SF=-1(force direction) → Accept**

Contact Condition

Since this model has a circumstance where two parts are in contact, we applied ***Contact_ Automatic_single_surface**, which enables simple and effective analysis. In this condition, there is no need to set separate slave and master elements. It is a relatively simple keyword because the program automatically finds and sets the contact surface. The coefficient of friction is set as 0.15 for both static and dynamic.

- **Keyword Manager → Contact → Automatic_Single_Surface → SSID, MSID =0 → FS=FD=0.15**



*CONTACT_AUTOMATIC_SINGLE_SURFACE_(ID/TITLE/MPP) (1)

☐ MPP1 ☐ MPP2

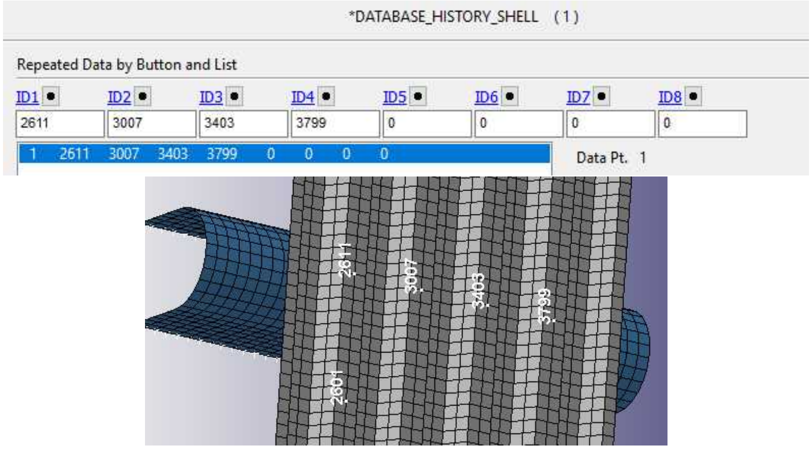
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSE	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	0	0	0	0	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.1500000	0.1500000	0.0	0.0	0.0	0	0.0	1.000e+20

▲ Contact condition setup

Analysis Setup

Since the event of the contact behavior can be captured from a run time of 0.01sec, set termination = 0.01 and set the **d3plot**, **ASCII** data, and **history shell** for extracting the working load in the output database.

- **Keyword Manager → Control | Termination → ENDTIM = 0.01**
- **→ ASCII_option → Default DT=0.0002, select GLSTAT, ELOUT**
- **→ BINARY_D3PLOT → NPLTC=40 (generate 40 frames)**
- **→ HISTORY_SHELL → select 4 nodes (see image)**



*DATABASE_HISTORY_SHELL (1)

Repeated Data by Button and List

ID1 ●	ID2 ●	ID3 ●	ID4 ●	ID5 ●	ID6 ●	ID7 ●	ID8 ●
2611	3007	3403	3799	0	0	0	0

```
1   2611   3007   3403   3799   0   0   0   0
```

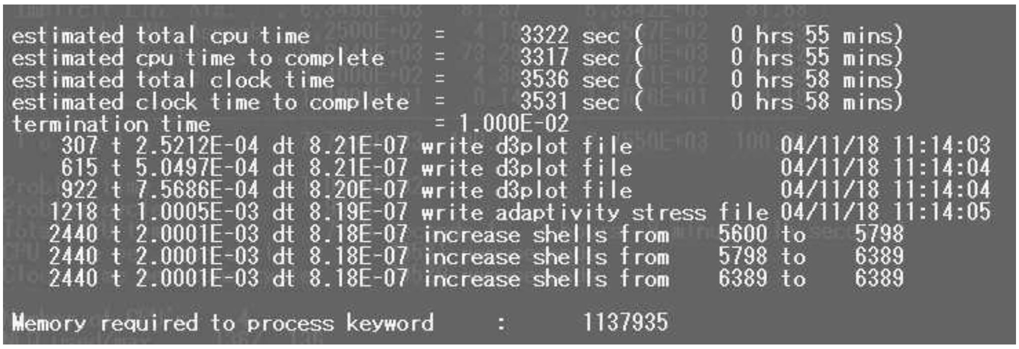
Data Pt. 1

▲ History nodes information

- → KEYWORD → MEMORY=800000000, NCPU=4 (if you have quad-core PC)
- • CONTROL → Contact → Default
- • CONTROL → Energy → HGEN=1, RWEN=1, SLNTEN=1, RYLEN=1
- CONTROL → ADAPTIVE → ADPFREQ=0.001, ADPTOL=0.2, ADPOPT=2, MAXLVL=3, IOFLAG=1

NOTE Here are the primary keywords for **ADAPTIVE Remehsing**. **ADPFREQ** is a cycle of regenerating the mesh. In this analysis, since 0.001 is set, 10 regeneration cycles occur. **ADPTOL** (Adaptive Error Tolerance_) means the error tolerance for remeshing and usually enters a value less than 2. The default value is 1e + 20 and the smaller the change, the smaller the element changes occur. (Adaptive option) is a method for the remeshing criteria, where EQ.1 varies depending on the angle change of the surrounding element and EQ2 changes the total angle change, and EQ.2 responds more sensitively. **MAXLVL** (maximum level of refinement) and flag to generate adaptive mesh at exit (**IOFLAG**) is an option to record the information of the generated mesh at the end of the analysis. Please note that the option to ADAPTIVE Remeshing occurs because ADPOPT is not enabled.

So far, we have generated keywords for a three-point bend test analysis model using an explicit solver. Compare the generated '*W7_SETUP_3P BendTest_explicit'* model with the generated model and see if there are any differences. Save the analysis model and run the analysis. Perform the analysis using **File → LS-Run of LS-Prepost**. In the case of the explicit model, the latest PC will complete the analysis within minutes. If the modeling was successful, the command prompt will show the information of adaptive remeshing as the analysis proceeds. As the mesh is subdivided like *'Shell increase from 5600 to 5798'*, the number of elements created by remeshing is displayed.

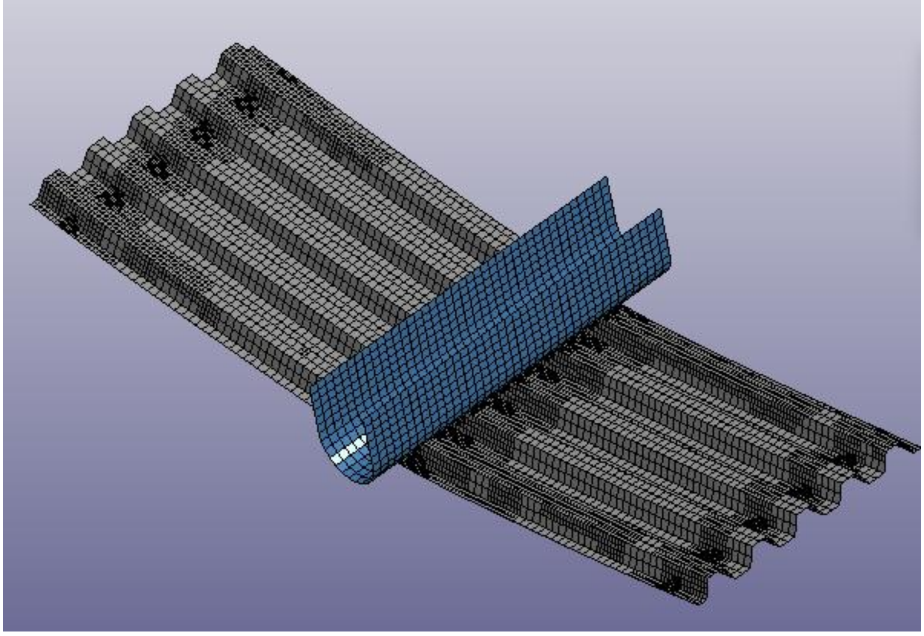


```
estimated total cpu time               =     3322 sec (       0 hrs 55 mins)
estimated cpu time to complete         =     3317 sec (       0 hrs 55 mins)
estimated total clock time             =     3536 sec (       0 hrs 58 mins)
estimated clock time to complete       =     3531 sec (       0 hrs 58 mins)
termination time                       = 1.000E-02
    307 t 2.5212E-04 dt 8.21E-07 write d3plot file              04/11/18 11:14:03
    615 t 5.0497E-04 dt 8.21E-07 write d3plot file              04/11/18 11:14:04
    922 t 7.5686E-04 dt 8.20E-07 write d3plot file              04/11/18 11:14:04
   1218 t 1.0005E-03 dt 8.19E-07 write adaptivity stress file 04/11/18 11:14:05
   2440 t 2.0001E-03 dt 8.18E-07 increase shells from     5600 to     5798
   2440 t 2.0001E-03 dt 8.18E-07 increase shells from     5798 to     6389
   2440 t 2.0001E-03 dt 8.18E-07 increase shells from     6389 to     6389

Memory required to process keyword     :       1137935
```

▲ Log screen

Result Visualization

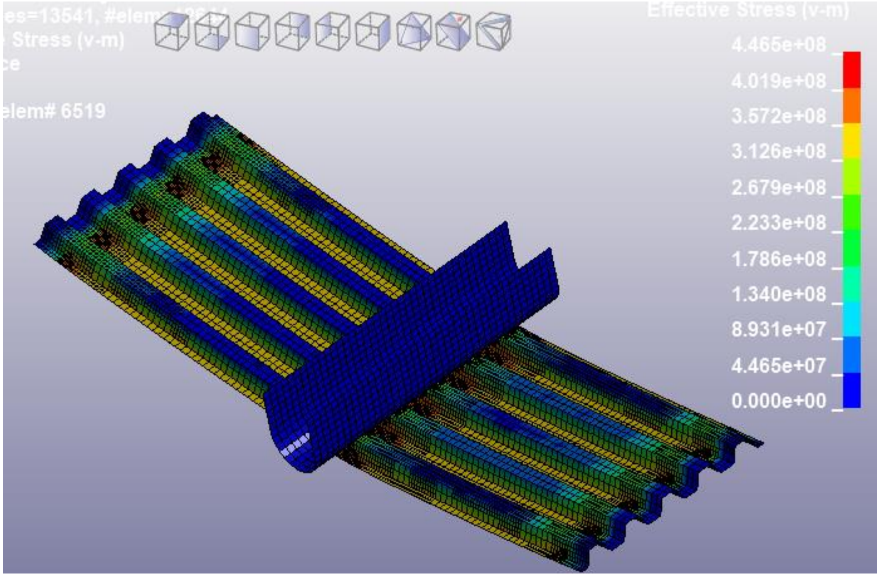
After the analysis is complete, load **D3plot** into Prepost. You can also load it directly by clicking the **LSPP** icon in the LS-Run dialog. Moving to the last frame, as shown in the figure below, it can be seen that many elements that are subdivided by adaptive remeshing are created. The result data shows that the number of elements starting with 5000 is more than doubled to about 12000 in the final frame.



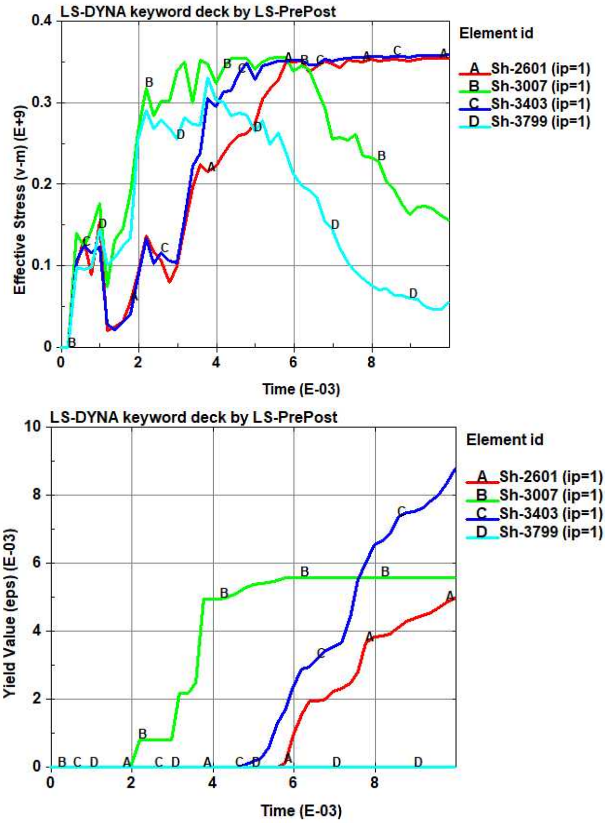
▲ Remeshed Corrugated Panel

If the von mises stress distribution is selected from the post fringe component, the von mises stress distribution can be seen as shown below. The maximum stress was about 440MPa which occurs at the region where bending deformation is maximum and the largest displacement was 50mm at the central section. When modeling the four nodes set in the history shell, select and plot four nodes in ASCII to extract the stress and the result data can be checked with the graph as shown below. With this data taken from Prepost and the data from the strain gage set in the experiment or the load data measured from the load cell can be compared with each other. If you click the **Save** icon below the graph, you can save it as various types of numerical data. If the **Yield value** (eps) is selected among the extraction data options, it is possible to check the timing of plastic deformation while yielding occurs. So far, we have practiced the analysis process with the explicit solver. Next, we will proceed with the same analysis by use of the implicit solver.

- **Post → Fringe component | Stress | Von mises stress**
- **Post → Fringe component | Ndv | Result displacement**
- **Post → ASCII → Elout* | Load → Sh-4 | Ip-1 or Ip-2 | Effective stress (v-m) → plot**



▲ Von mises stress distribution



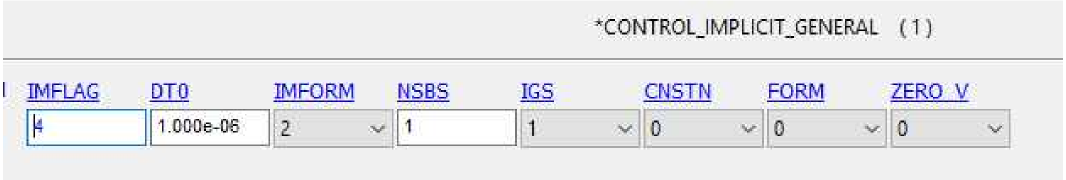
▲ Stress distribution and Yield stress extracted from center 4 nodes (Explicit solver)

B. Implicit Solver Model

Implicit solver setup

Save the generated model with a new name and edit the keywords. Add two more keywords, ***Implicit_General** and ***Implicit Solution**, as shown below.

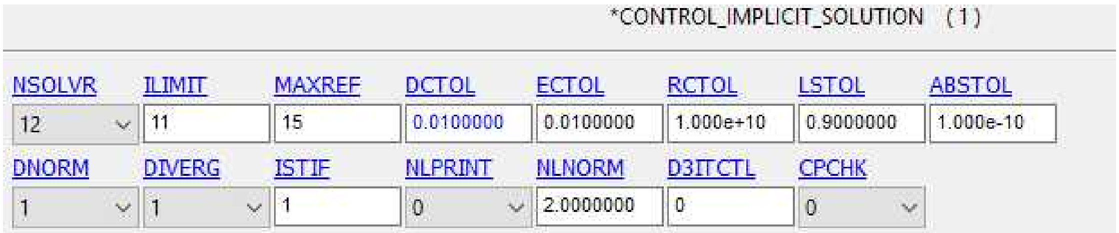
- **Keyword Manager → Control → Implicit_General → IMFLAG = 4 | DT0 = 1e-6 | IMFORM = 2 | Enter IGS = 1 → Accept**
- **Keyword Manager → Control → Implicit_Solution → Default Value → Accept**



*CONTROL_IMPLICIT_GENERAL (1)

IMFLAG	DT0	IMFORM	NSBS	IGS	CNSTN	FORM	ZERO_V
4	1.000e-06	2	1	1	0	0	0

▲ Implicit General keyword generation



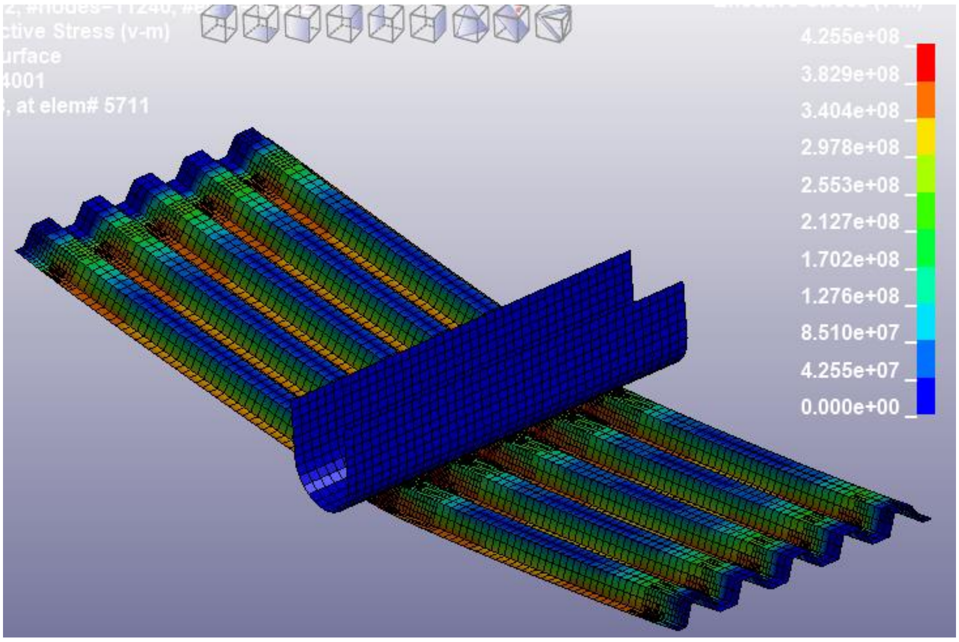
*CONTROL_IMPLICIT_SOLUTION (1)

NSOLVR	ILIMIT	MAXREF	DCTOL	ECTOL	RCTOL	LSTOL	ABSTOL
12	11	15	0.0100000	0.0100000	1.000e+10	0.9000000	1.000e-10
DNORM	**DIVERG**	**ISTIF**	**NLPRINT**	**NLNORM**	**D3ITCTL**	**CPCHK**	
1	1	1	0	2.0000000	0	0	

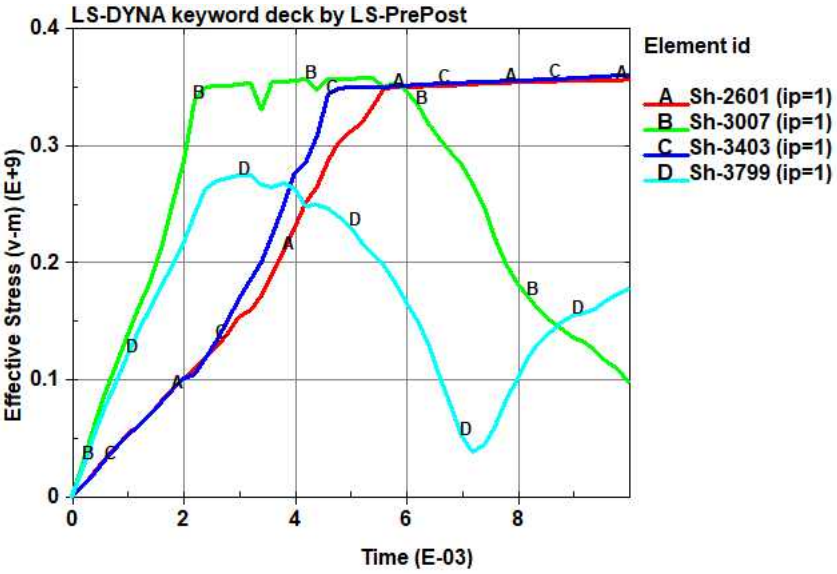
▲ Implicit Solution keyword generation

Create two additional keywords like this and proceed with the analysis. Perhaps an error will occur after a few minutes of analysis at a fairly slow pace. The error message says that it failed to find convergence, which is one of the most important differences between implicit and explicit. In order to find an equilibrium condition for implicit convergence, many requirements must be met.

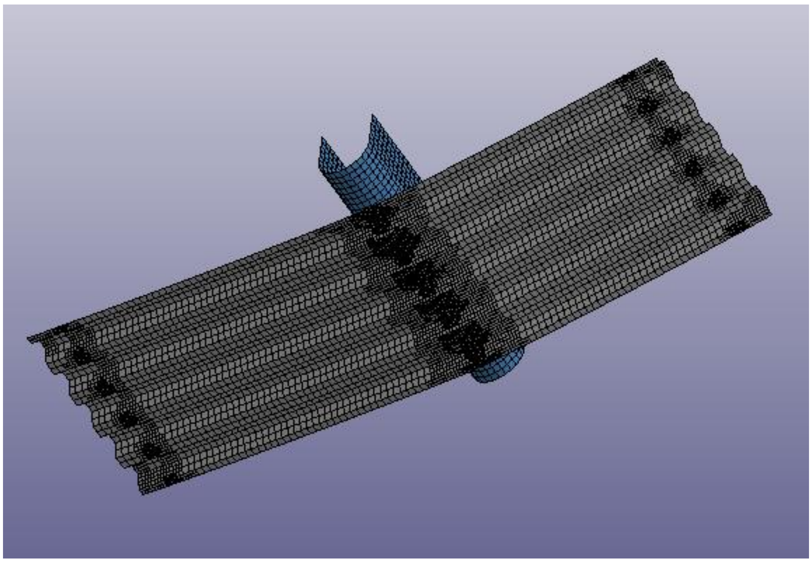
Now, let's rerun the analysis after increasing the **DCTOL** variable 10 times from 0.001 to 0.01. The **DCTOL** variable is a value of Displacement convergence tolerance, which means that the numerical criterion is more relaxed. You can see that the step is progressing a little faster than before. The analysis will be completed in about 2-3 hours without any error message. Compared to an explicit model completed in minutes, the implicit analysis takes a much longer time to complete. The maximum stress is 425 MPa, which is slightly different from the explicit 446 MPa. Let's compare the stress distribution graph extracted from the node in the center with the remeshed state. In the case of explicit analysis, the change of stress is more significant and the remeshed state shows a more uniform in implicit case, whereas it is more scattered in Explicit.



▲ Von mises stress distribution (Implicit model)



▲ Stress distribution extracted from center 4 nodes (Implicit solver)



▲ Remeshed Corrugated Panel (Implicit solver)

We have analyzed the analysis of the 3Point bending test for the corrugated panel by the implicit and explicit method and also compared the differences. These differences are typical factors of the two solvers. We also practiced how to apply **Adaptive Remeshing**. Let's see how it affects the results of the analysis by changing parameters related to **Adaptive Remeshing**. The best way to understand parameter is to learn from these various attempts and iterations carried out with a different value of any parameter.

Difference between Implicit and Explicit

Accurately understanding the difference between explicit and implicit is an important concept as it is linked to how to interpret the target structure in an effective way. However, many engineers in the field simply understand whether the variation over time is large or small. Thus, in general, collision analysis is often explicit and general mechanical behavior analysis is implicit. This is not a completely wrong case, but it is necessary to derive more scientific analysis through understanding the underlying concept of implicit and explicit analysis scheme.

Although it is known as a dynamic analysis program such as LS-DYNA and Autodyn, all include implicit solvers and general programs such as ABAQUS, ANSYS, and NASTRAN also include both implicit and explicit solvers.

Implicit solver

The following formulas dealt with in FEM class basically implicit interpretation concept. ***[K]*** stands for the stiffness matrix, ***[U]*** stands for displacement matrix, and ***[F]*** stands for load matrix.

$$[K]\,[U] = [F]$$

It is a system equation consisting of a large matrix that is expressed as a whole system by combining the basic equations generated from the individual elements to be analyzed. This system equation is created through a combination of the number of elements in the system considering contact conditions, boundary conditions, and load conditions, etc. In Implicit, the solution is solved by solving the simultaneous equation for each step to find the exact solution. The equilibrium state inside the system with respect to external loads must be maintained in order to solve the problem. If the system's nonlinearity is high, it may not converge and the solution of the simultaneous equations may not be calculated and thus the analysis may not be completed. These solutions are unconditionally stable and facilitate larger time steps. Despite this advantage, the implicit methods can be extremely time-consuming when solving dynamic and nonlinear problems.

Explicit solver

The explicit solver does not solve the stiffness matrix but solves the solution by substituting the value of the previous step into the next step. Therefore, there is no burden of convergence because the solution to the system of equations is not solved at each stage and the solution may have a different accuracy but a result can be obtained in any way. Therefore, in order to increase the accuracy of the analysis, the size of the time step must be reduced to increase the accuracy. In other words, the higher the nonlinearity is, the smaller the time step can be to obtain meaningful results. To be more precise, the time step in an explicit finite element analysis must be less than the Courant time step (i.e., the time taken by a sound wave to travel across an element) while implicit analyses have no such limitations. It is the analysis engineer who needs to consider the analysis time and the applicability of the analysis result.

As can be seen from the results of this workshop, the difference between the results of implicit and explicit occurs within reasonable accuracy, but the analysis time is two to three hours in the case of implicit while a couple of minutes in the explicit analysis. Thus, how to solve a problem effectively depends on the engineer's experience and knowledge.

Compare the differences between Implicit Solver and Explicit Solver

All of these problems are expressed through mathematical partial differential equations (PDE’s). Implicit is a method of solving the system of equations of the next step (t_{n+1}) through the solution of the system of equations of the corresponding step (t_n), while explicit uses the state of the step (t_n) in the next step. The more effective solver should be selected according to the FE model. Implicit is excellent in the static load analysis, buckling, spring back, and thermal analysis, while the explicit is absolutely excellent high nonlinear collision analysis and explosion analysis. For dynamic problems, the matrix equations come down to:

$$[M]\{x''\} + [C]\{x'\} + [K]\{x\} = \{f\}$$

Where ´ and ´´ represent the first and second derivatives.

- **Explicit governing equation**

$$x_{n+1} = f(x_n, x_{n-1}, \dots, t_{n+1}, t_n, \dots)$$

$$[M]\{x''\}+[C]\{x'\}+[K]\{x\}=\{F\}$$

- **Implicit governing equation**

$$x_{n+1} = f(x_{n+1}, x_n, x_{n-1}, \dots, t_{n+1}, t_n, \dots)$$

$$[M]\{x''\}+[C]\{x'\}+[K]\{x\}=\{F\}$$

Calculate x” directly in the system of equations. So, the Inverse of the stiffness matrix should be calculated.

Implicit	Explicit
▪ For each step, the solution to the stiff matrix system is to be solved. ▪ In the case of static load, a very large step size may be applied compared to explicit. ▪ The system remains stable even when relatively large step size is applied, which is called an unconditionally stable scheme. ▪ As the system must always be balanced, the load or boundary conditions must be accurately represented. ▪ In case of large nonlinearity, not only the analysis takes a considerable amount of time, but also the error occurs due to the lack of convergence. ▪ Applied to the problem of long-time phenomenon like static / dynamic behavior analysis	▪ In case of high nonlinearity, nodal acceleration is calculated directly, so the computational efficiency is excellent. ▪ To improve the accuracy of the analysis, very small-time steps should be applied. ▪ Since the inverse matrix is not calculated, relatively small memory is used. ▪ It is suitable for solving the problem of high nonlinearity dynamics. ▪ It is effective for a system having many contacts. ▪ Applied to short-time dynamics problems that occur in a short time such as impact and crash analysis

When to use Explicit or Implicit FEM?

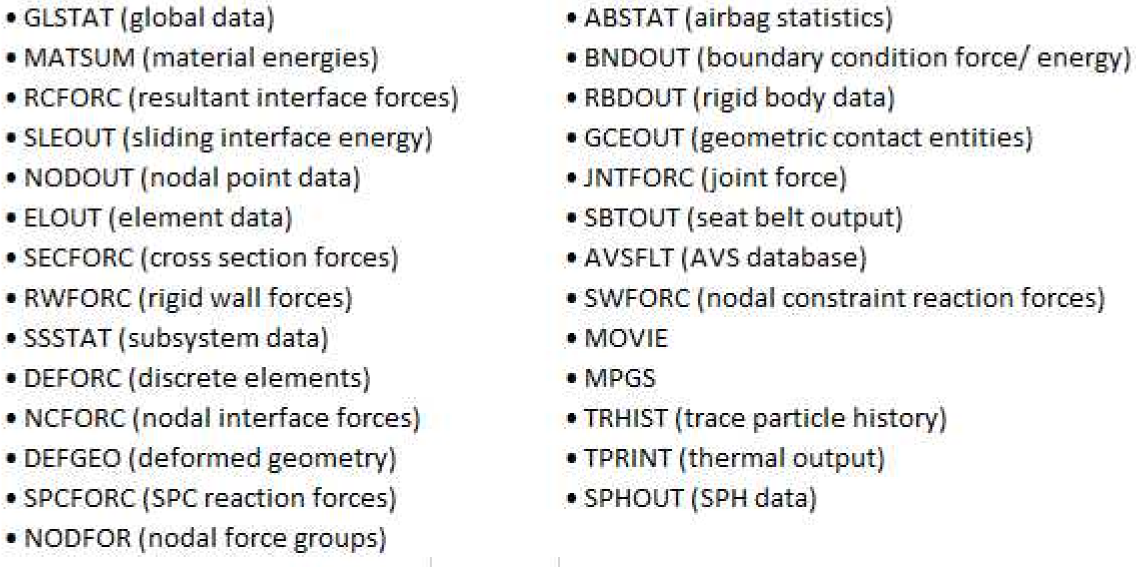
The explicit analysis offers a faster solution in events where there is a dynamic equilibrium or otherwise: Sum of all forces = mass x acceleration. The explicit method should be used when the strain rates/velocity is over 10 units/second or 10 m/s respectively. These events can be the best example of extreme scenarios such as an automotive crash, ballistic event, or even meteor impact. In these cases, the material models do not only need to account for the variation of stress with strain but also the strain rate. On this scale, the strain rates play a particularly important contribution. The implicit method should be used when the events are much slower and the effects of strain rates are minimal. Once the growth of stress as a function of strain can be established, these can be analyzed using implicit methods. In this case, one can consider a static equilibrium such that: Sum of all forces = 0. This covers many of the most common engineering problems.

Knowledge Output files

The result files produced by LS-DYNA analysis are divided into binary file type and ASCII file type. Binary files are data that can be displayed as images during the post-processing of Prepost, and ASCII files are output data that can be displayed as graphs by outputting numerical data. All can be controlled via the keyword * **DATABASE_**<option>. Results from various controls are recorded in the **d3hsp** file and modeling data and error messages are recorded in the message file. The types according to the extension of the output file are as follows.

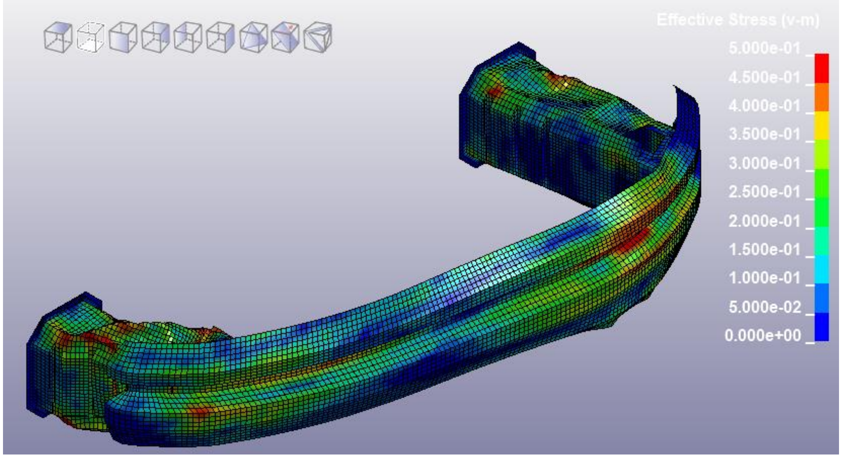
- **D3PLOT:** Binary format data where most of the analysis results contain data. This is the most important and large file to check when displaying results in Prepost.
- **D3DUMP:** Binary file that records all data for restart
- **RUNRSF:** Data required for restart, file overwriting the latest data repeatedly. * Restart requires either D3DUMP or RUNRSF**.**
- **D3DRLF:** Database for Dynamic Relaxation
- **Dynamic Relaxation:** One of the numerical techniques that lead to an equilibrium state by applying initial load and boundary conditions to the target system. In FEM, oscillation such as vibration occurs when the initial load is applied to the system, and the process of reaching equilibrium with time is called Dynamic Relaxation (DR) in LS-DYNA.
- **D3MEAN:** Database for CFD Analysis
- **INTFOR:** Database of results generated at the contact surface
- **XTFILE:** Data about extra time history
- **D3EIGV:** File that modal data is saved when analyzing Eigenvalue
- **D3CRCK:** Results data file for crack growth when using Winfrith concrete model

The following shows the meaning of the ASCII result file. There are so many variables that I'm not sure at first what to apply, but there are only a few commonly used parameters. **GLSTAT, MATSUM, RCFORC**, and **ELOUT** are the most frequently used parameters. Let's look at the meaning of each parameter and learn one by one according to the type of interpretation.



- GLSTAT (global data)
- MATSUM (material energies)
- RCFORC (resultant interface forces)
- SLEOUT (sliding interface energy)
- NODOUT (nodal point data)
- ELOUT (element data)
- SECFORC (cross section forces)
- RWFORC (rigid wall forces)
- SSSTAT (subsystem data)
- DEFORC (discrete elements)
- NCFORC (nodal interface forces)
- DEFGEO (deformed geometry)
- SPCFORC (SPC reaction forces)
- NODFOR (nodal force groups)
- ABSTAT (airbag statistics)
- BNDOUT (boundary condition force/ energy)
- RBDOUT (rigid body data)
- GCEOUT (geometric contact entities)
- JNTFORC (joint force)
- SBTOUT (seat belt output)
- AVSFLT (AVS database)
- SWFORC (nodal constraint reaction forces)
- MOVIE
- MPGS
- TRHIST (trace particle history)
- TPRINT (thermal output)
- SPHOUT (SPH data)

Chapter 8. Impact Analysis of Crush Box



Objectives

- ✓ **Understanding Structural Characteristics of Vehicle Crush Box**
- ✓ **Practicing the procedure for modifying elements**
- ✓ **Assembling Multiple Parts (* Contact_Tied)**
- ✓ **Learning the analysis results visualization**

Duration: 80min

Application: Auto, Explicit, Energy absorption

Model files

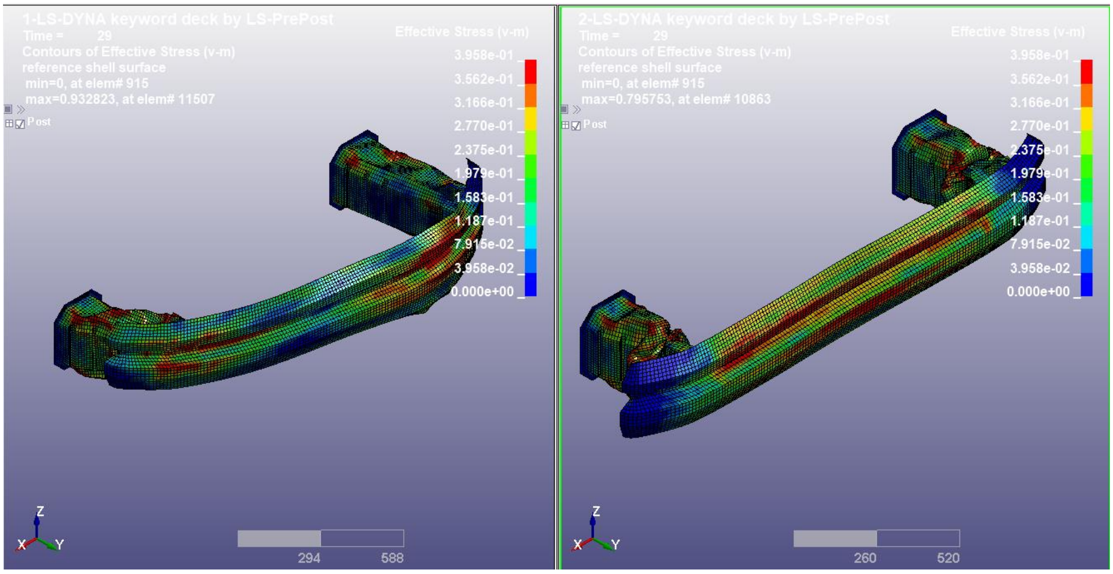
- W8_CrushBox.bdf: Meshed Nastran format model
- W8_Initial_Crushbox_mesh.k: Initial state model (Mesh)
- W8_SETUP_Crushbox.k: Final state model (Normal mesh)
- W8_SETUP_Crushbox.k_Refined.k: Final state model (Refine mesh)
- W8_SETUP_Crushbox.k_Refined_angled.k: Final state model (inclined impact)

W8

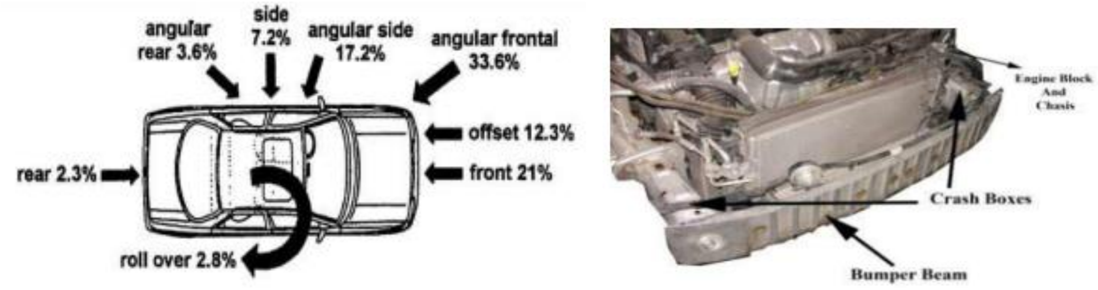
Workshop 8.
Impact analysis of Crush Box

Introduction

In passenger vehicle design, one of the most significant factors in determining passenger safety is the development of the frontal shock absorbing structure. LS-DYNA is widely used in the design of such shock absorbing structures. In this exercise, we shall practice the impact analysis of the front crush box structure and perform the steps of importing the shape model, keyword modeling in LS-Prepost and finally reviewing the analysis results. The crush box of a car absorbs the impact energy generated during the impact of the car and serves as a structural material to alleviate the impact energy imparted to the chassis or passengers.

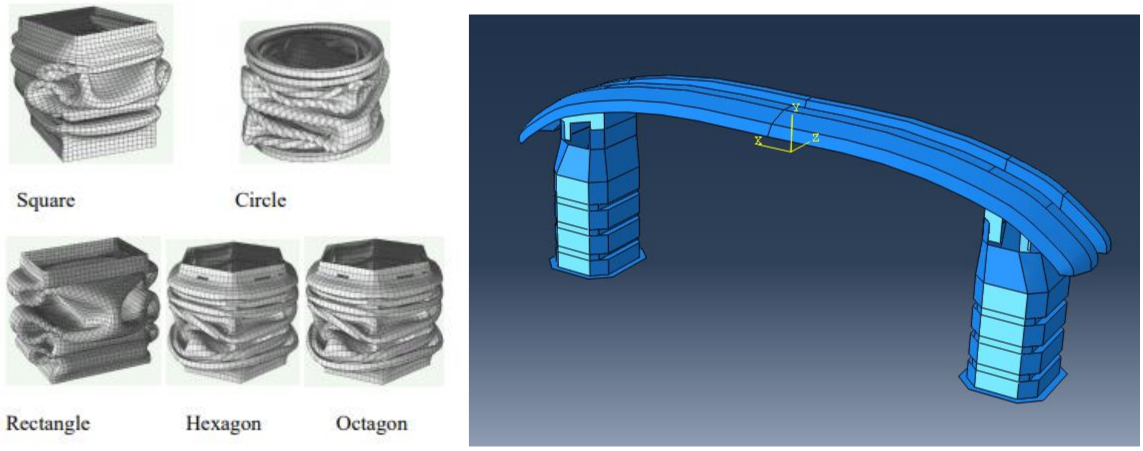


▲ Analysis results of the crush box assembly



▲ The ratio of impact direction of vehicle and mounting position of the crush box

In general, a crush box can be designed having a rectangular, circular, hexagonal, octagonal shape that is mounted on a vehicle. Here the octagonal shape is known to have an excellent energy absorption rate. FEM analysis is widely applied for optimal shape design for energy absorption and the main energy absorption mechanism absorbs the impact energy by the folding mechanism (crumpling) which distorts while undergoing plastic deformation. Various shape design elements are applied to create such folding. In this exercise, we will analyze using a crush box design in which the bead initiator is designed with an octagon shape.



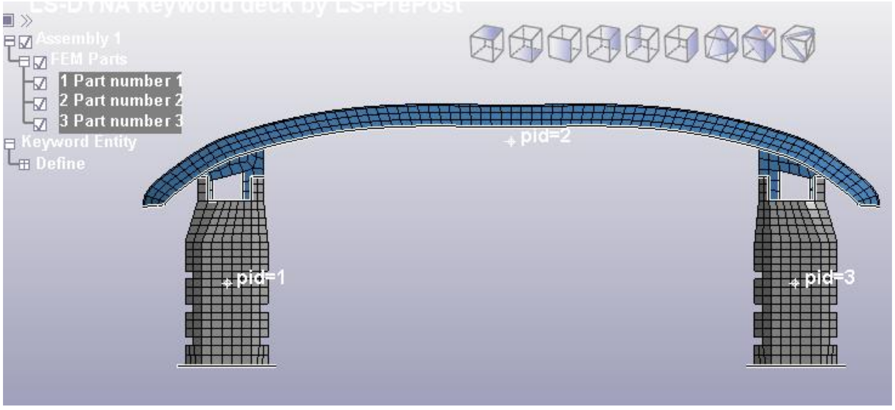
▲ Various Crush Box Shapes and Practice Models

Import Mesh Model

If you import the provided Nastran model file ***'W8_Crashbox.bdf'*** from Prepost, you will see a three-part model with elements as shown below. Although the general unit of mechanical components uses the meter system, the following system is used in this workshop:

- **Unit system [mm, kg, ms, kN, GPa, kN-mm]**

Let's look at the imported model. The model consists left and right crush box with a bump beam joined together and you can check the section information with the element information that is imported together. Of these, only shape information including elements will be used and the rest of the modelization will be modeled by generating new keywords in LS-Prepost.



▲ Imported mesh model

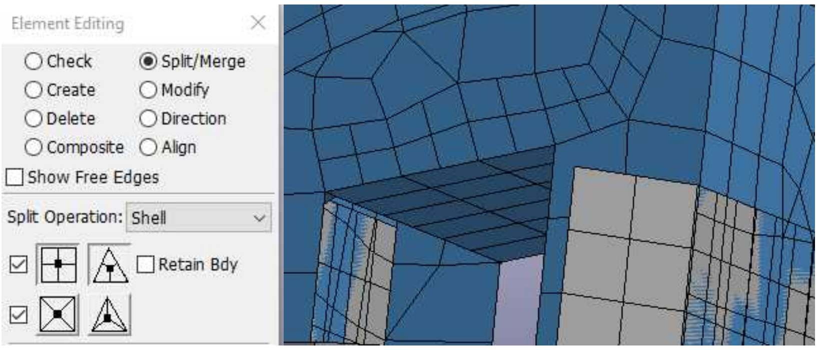
Divide Element

This model has meshed with good-quality elements, but we will work further to refine the elements that are relatively large in size. A good element is an element having a good aspect ratio that does not distort much of the horizontal/vertical axis.

Use the following option to refine the mesh:

- **Element tools → Element editing → Split/Merge**
- **Sel. Shells tool box → Pick | Add | By elem → select one element → Apply | Accept**

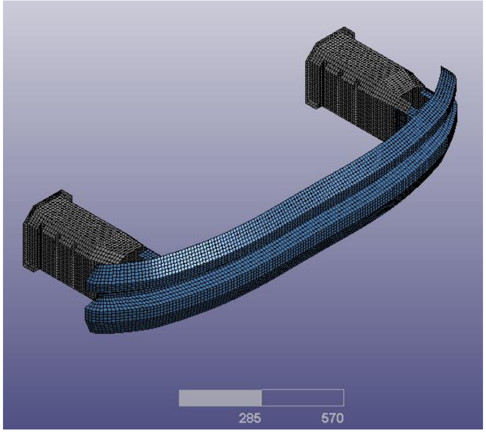
You can see that one element is divided into four as shown below. In this way, the mesh can be further refined. Select the whole to refine the mesh.


Element Editing
○ Check ● Split/Merge
○ Create ○ Modify
○ Delete ○ Direction
○ Composite ○ Align
☐ Show Free Edges
Split Operation: Shell
☑ ☐ Retain Bdy
☑

▲ Element splitting

- **Sel. Shells tool box → Pick | Area | By elem → Select whole model → Apply | Accept**

  * Please note that the entire element of a part should be divided in order to get elemental connectivity. Else there shall be a discontinuity between the fine element and the coarser element which cause an abrupt change in stress or displacement values



▲ Model remeshed by element splitting

Material Property

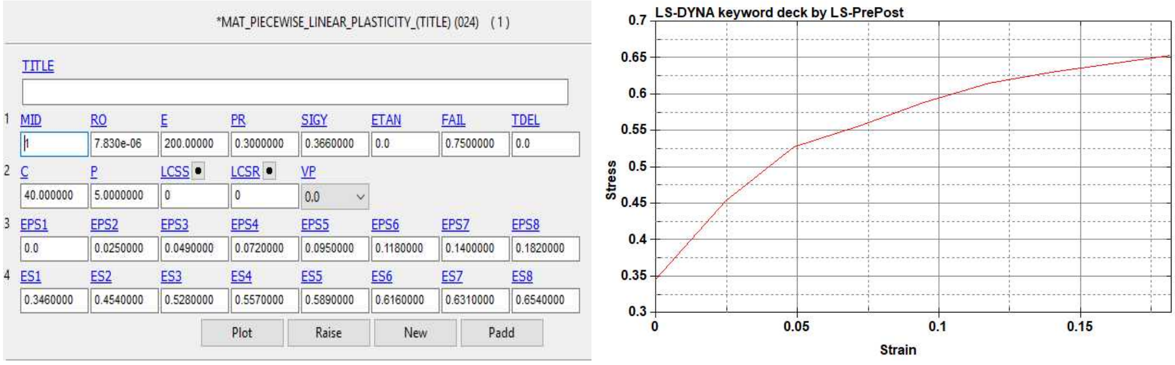
Crush box components are made of stainless steel and are modeled using the **MAT24 MAT_PIECEWISE_LINEAR_PLASTICITY** keyword, an elastic-plastic model. It is a model that can input the physical properties of the plastic zone in tabular form. If you click Plot after input, you can check the input value of the plastic area by the graph.

- **Keyword Manager → MAT | 024-Piecewise_linear_plasticity → Input properties**

NOTE Enter the property values by paying attention to the unit system. Since the unit of stress is GPa, enter Young's modulus 200GPa of stainless steel as 200 and the density of 7850kg/m^3 as 7.85e-6. A strain is a dimensionless unit, so simply enter its value.

- **Keyword Manager → SECTION | Shell → ID=1, ELFORM=2, SHRF=1, NIP=3, T1=T2=T3=T4=1.0**
- **Repeat generation of 2 Shell section → SECTION | Shell →ID=2, ELFORM=2, SHRF=1, NIP=3, T1=T2=T3=T4=2.0**

NOTE ELFORM = 2 is a 4 node Belytschko-Tsay cell model that can be widely used as a general structural shell element. NIP is the number of integration points, the number of integration points in the shell's thickness direction.

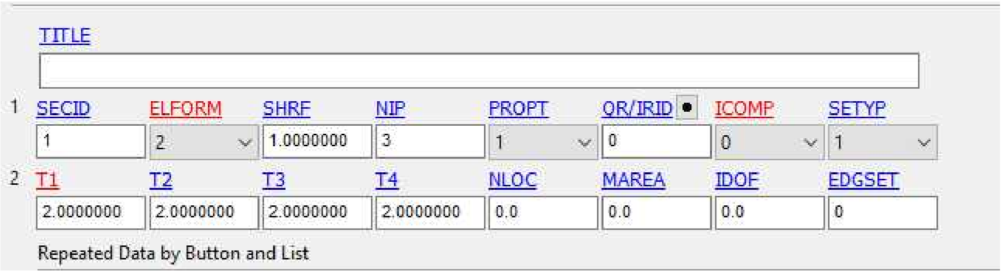


*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE

1	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
	1	7.830e-06	200.00000	0.3000000	0.3660000	0.0	0.7500000	0.0
2	C	P	LCSS ●	LCSR ●	VP			
	40.000000	5.0000000	0	0	0.0			
3	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
	0.0	0.0250000	0.0490000	0.0720000	0.0950000	0.1180000	0.1400000	0.1820000
4	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
	0.3460000	0.4540000	0.5280000	0.5570000	0.5890000	0.6160000	0.6310000	0.6540000

▲ Material Properties (MAT24) setup and plastic region graph



TITLE

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
	1	2	1.0000000	3	1	0	0	1
2	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
	2.0000000	2.0000000	2.0000000	2.0000000	0.0	0.0	0.0	0

Repeated Data by Button and List

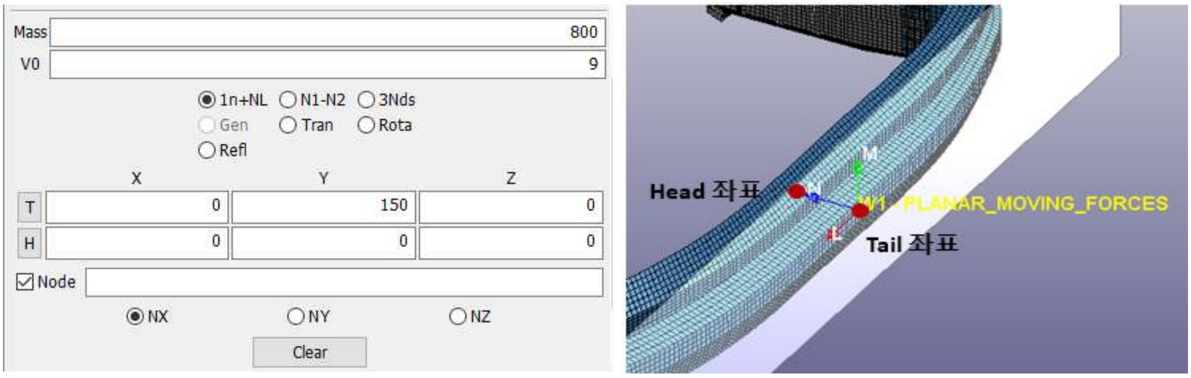
▲ Section properties

Boundary Condition

The boundary condition is as follows : fix the bottom of the crush box and since it collides with the rigid plane at 9m / s, The rigid wall velocity movement is set by entering the start vector (**Tx, Ty, Tz**) and the end position (**Hx, Hy, Hz**) for the plane vector. The size of the rigid wall is automatically set by Prepost considering the maximum and minimum size of the model.

- **Create Entity → Rigid Wall | Select Cre → Mass = 900, Vo = 9 Input → Done**
- **Create Entity → SPC | Select Cre → Select | Pick | By element | Prop, angle = 5 → box bottom selection → check X, Y, Z, RX, RY, RZ (111111) → Done**

In the case of the rigid wall for the 45-degree slope collision analysis, it can be created by changing the coordinates of the normal vector.



Mass 800

V0 9

● 1n+NL ○ N1-N2 ○ 3Nds

○ Gen ○ Tran ○ Rota

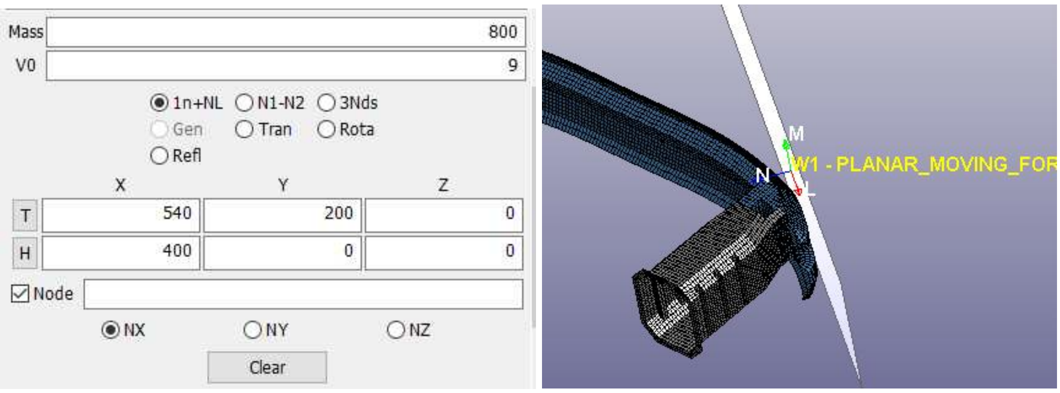
○ Refl

	X	Y	Z
T	0	150	0
H	0	0	0

☑ Node

● NX ○ NY ○ NZ

▲ Rigid Plane Generation (Frontal impact)



Mass 800

V0 9

● 1n+NL ○ N1-N2 ○ 3Nds

○ Gen ○ Tran ○ Rota

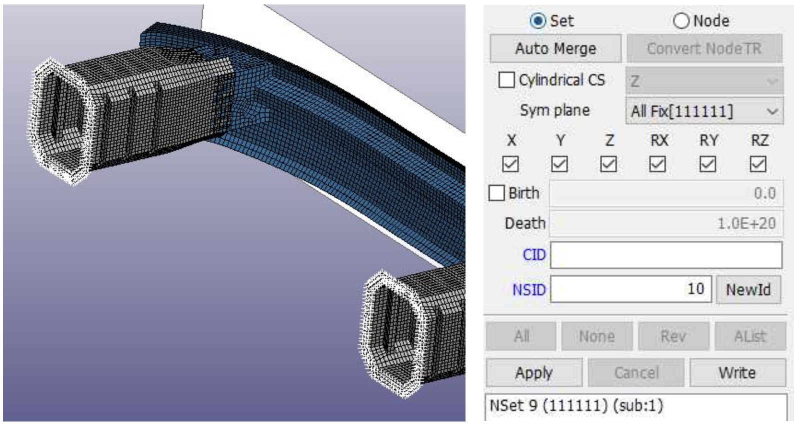
○ Refl

	X	Y	Z
T	540	200	0
H	400	0	0

☑ Node

● NX ○ NY ○ NZ

▲ Rigid Plane Generation (Side impact)



● Set ○ Node

☐ Cylindrical CS Z

Sym plane All Fix[111111]

X	Y	Z	RX	RY	RZ
☑	☑	☑	☑	☑	☑

☐ Birth 0.0

Death 1.0E+20

CID

NSID 10

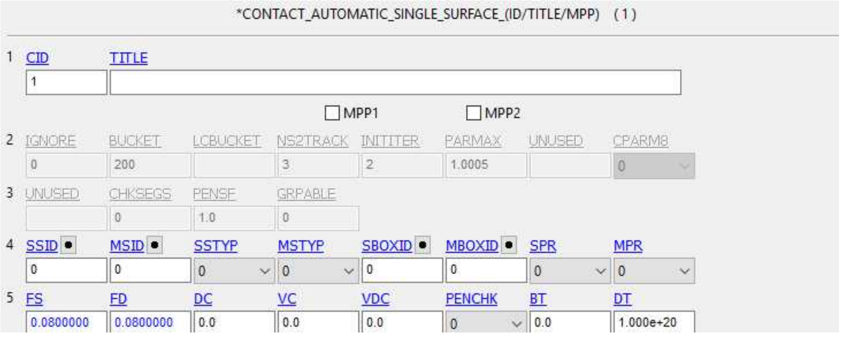
NSet 9 (111111) (sub:1)

▲ Boundary condition setup

Contact Condition

For contact conditions, it is better to use ***Automatic_single_surface** when using rigid walls. There is no need to set up a separate master surface or slave surface. DYNA automatically selects and applies the surface that comes into contact with the rigid wall.

- **Keyword Manager → Automatic_single_surface selection → SSID, MSID = 0, Friction Coefficient FS = FD = 0.08 → Done**



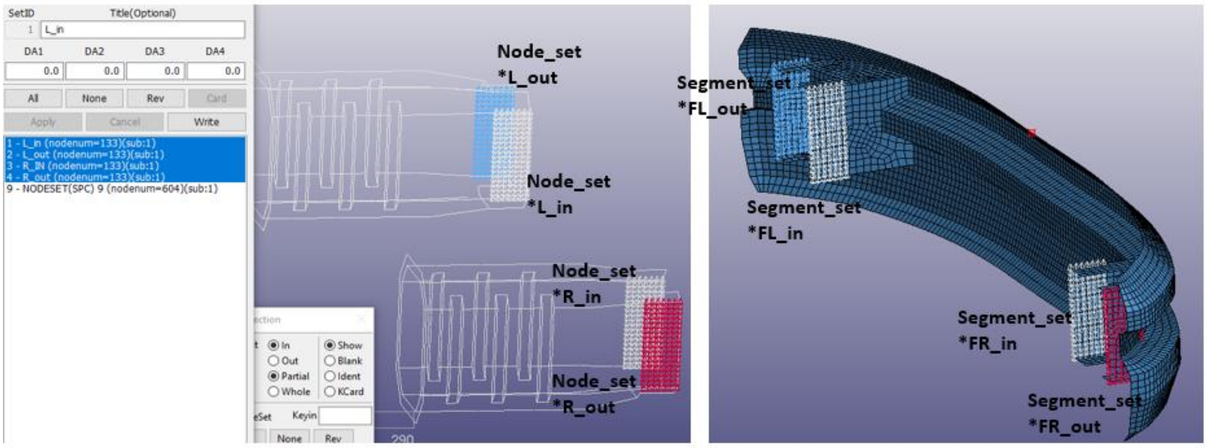
*CONTACT_AUTOMATIC_SINGLE_SURFACE_(ID/TITLE/MPP) (1)

1	CID	TITLE						
	1							
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSE	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	0	0	0	0	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.0800000	0.0800000	0.0	0.0	0.0	0	0.0	1.000e+20

▲ Surface contact condition setup

Select and input **Tied_Nodes_to_surface** in Contact condition to connect the Bumper beam and Crush box. Note that one side must define **Node_set** and the other side must define **Segment_set** when setting. In order to make a set like this, associate the set part as shown below.

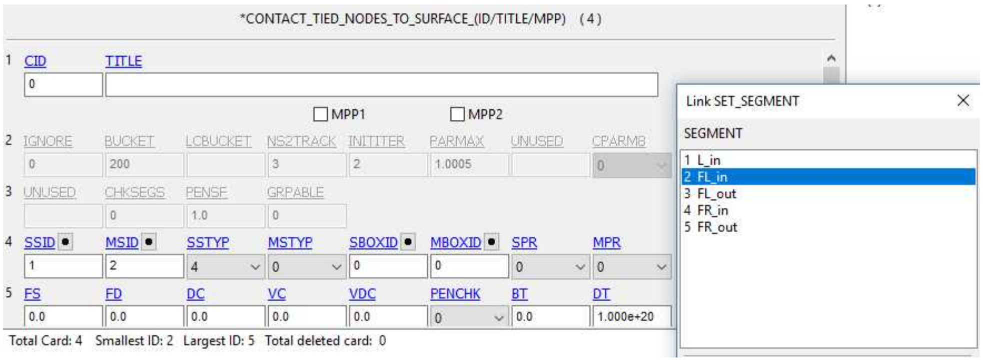
- **Create Entity → Set Data | Select Cre → Create Set_Node, Set_Segm (see below) → Done**
- **Keyword Manager → Contact_tied_nodes_to_surface → SSTYP = 4 (node set), MSTYP = 4 (segment set) and select the corresponding set to create 4 conditions → Done**



SetID	Title(Optional)		
1	L_in		
DA1	DA2	DA3	DA4
0.0	0.0	0.0	0.0

1 - L_in (nodenum=133)(sub:1)
2 - L_out (nodenum=133)(sub:1)
3 - R_IN (nodenum=133)(sub:1)
4 - R_out (nodenum=133)(sub:1)
9 - NODESET(SPC) 9 (nodenum=604)(sub:1)

▲ Set node and segment setup



*CONTACT_TIED_NODES_TO_SURFACE_(ID/TITLE/MPP) (4)

1	CID	TITLE						
	0							
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSE	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	1	2	4	0	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.0	0.0	0.0	0.0	0.0	0	0.0	1.000e+20

Total Card: 4 Smallest ID: 2 Largest ID: 5 Total deleted card: 0

Link SET_SEGMENT

SEGMENT

1 L_in
2 FL_in
3 FL_out
4 FR_in
5 FR_out

▲ Tie contact setup

Part property

This step helps to assign the properties and section properties created so far in the Part.

- **Keyword Manager → Part → LBOX and RBOX (T = 1.0 select shell section), Front Bumper select T = 2.0 shell section → Apply the same MAT24**

Analysis setup

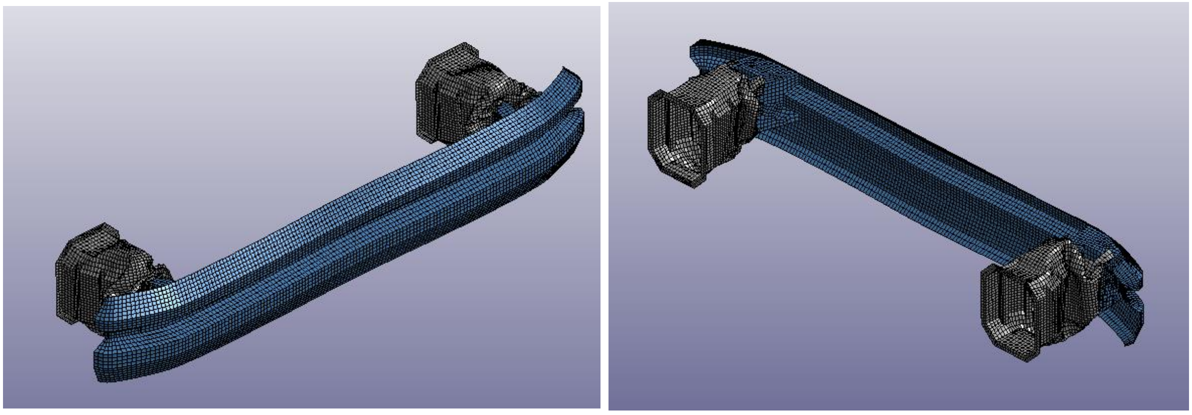
The analysis is requested to generate 30 D3PLOTs for a termination time of 30ms.

- **Keyword Manager → Control | Termination → ENDTIM = 30**
- **→ ASCII_option → Default DT = 0.1 and input GLSTAT, MATSUM, NODOUT, RWFORC among various parameters**
- **→ BINARY_D3PLOT → DT = 1 input**
- **→ BINARY_D3THDT → DT = 0.1 input**
- **→ KEYWORD → MEMORY = 800000000, NCPU = 4 (enter according to user's environment)**

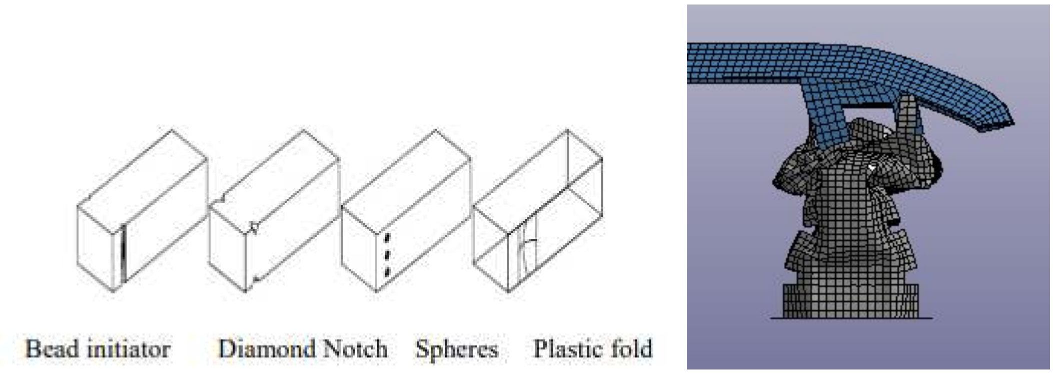
The generated model is then compared with the provided completed file ‘***W8_SETUP_Crushbox.k’*** keyword. There may be a difference between the **MATID** and **SECID** entered by the readers.

Analysis result

Run the analysis and load the result file **D3PLOT** from the prepost. The **D3plot** for 30 frames is created and shows the deformation as shown below. Crush box absorbs a lot of impact energy through the crushing process while undergoing plastic deformation. There are several design elements to induce this folding. The crush box applied in this exercise can also be seen that the compression deformation is caused by rapid folding around the bead initiator.



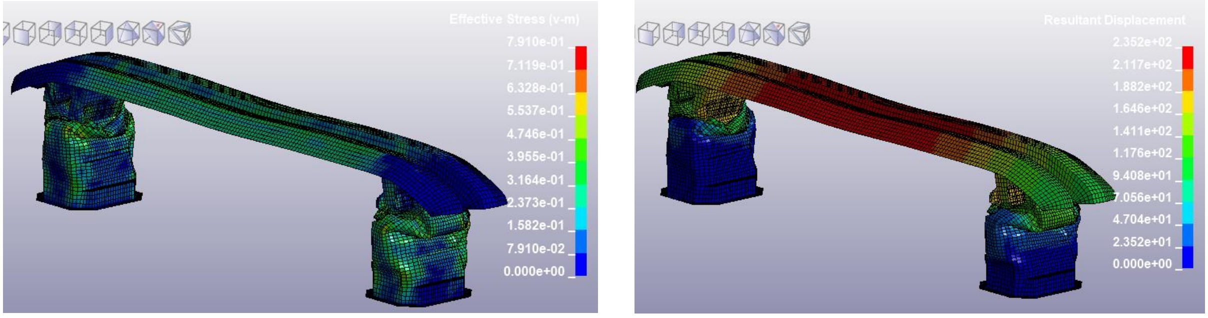
▲ Deformation shape



▲ Folding trigger design factor and compression deformation shape

If the von mises stress distribution is selected from the post fringe component, the von mises stress distribution can be seen as shown below. If you want to check the stress concentration area more clearly, you can lower the maximum stress slightly in the Fringe range option to more easily identify the areas where the stress is large. In the deformation, the largest displacement of 235mm occurred at the center of the bumper beam and the maximum stress occurred was 791MPa.

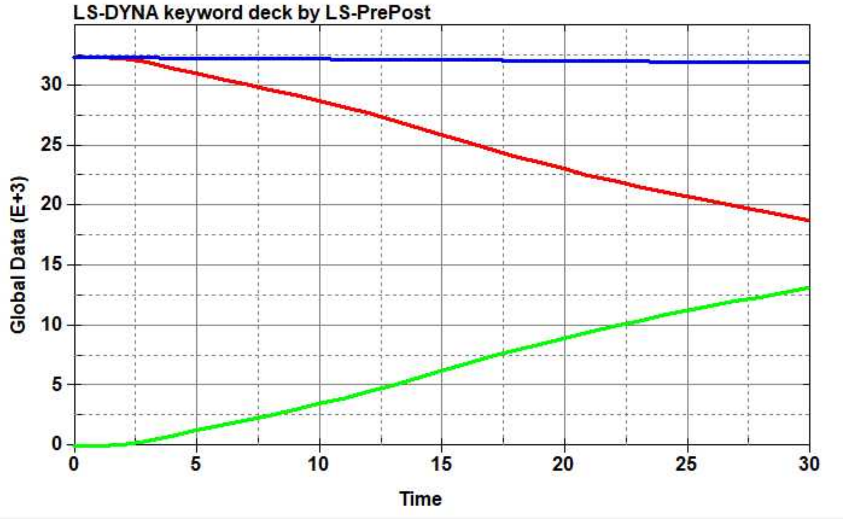
- **Post → Fringe component | Stress | Von mises stress**
- **Fringe range → User | Active Part Only | Max = 0.5**
- **Post → Fringe component | Ndv | Result displacement**



▲ Von mises stress distribution and Result displacement

Care should be taken when analyzing the results. Since the unit of stress is GPa, the result of 7.91e-1 should be converted to 791MPa and 2.35e+2 to 235mm. If you check the energy distribution occurring in the whole model, it is distributed as shown in the figure below. The total distribution shows the total energy, the internal strain energy which gradually increases in the number and the kinetic energy, decreasing gradually. The energy absorption effect of the target structure of the collision analysis can be analyzed through the energy distribution. In general, the graph is displayed with a black background. If you select **Attribute → Invert** at the bottom of the plot, you can get a plot with a white background. This is an easy option when printing.

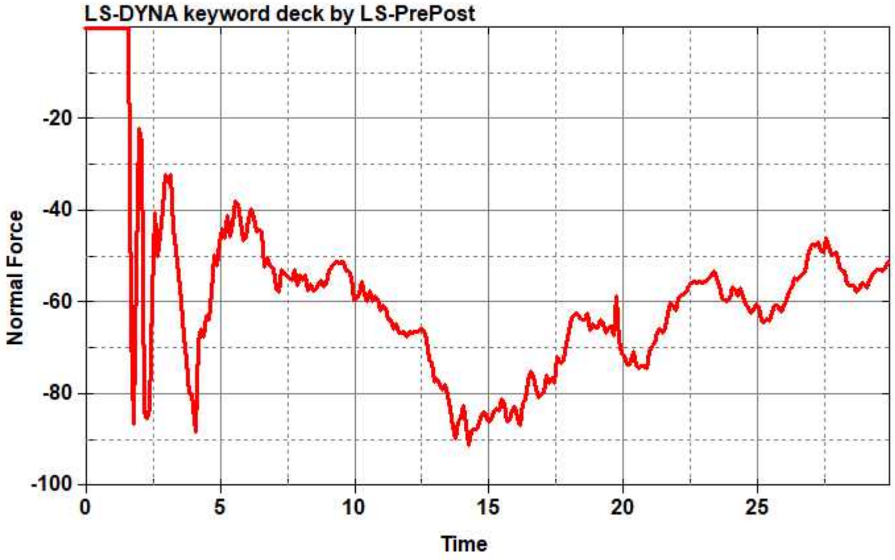
- **History → Global | Select all Kinetic, Internal, Total energy → Plot**



▲ Kinetic, Total, Internal energy distribution

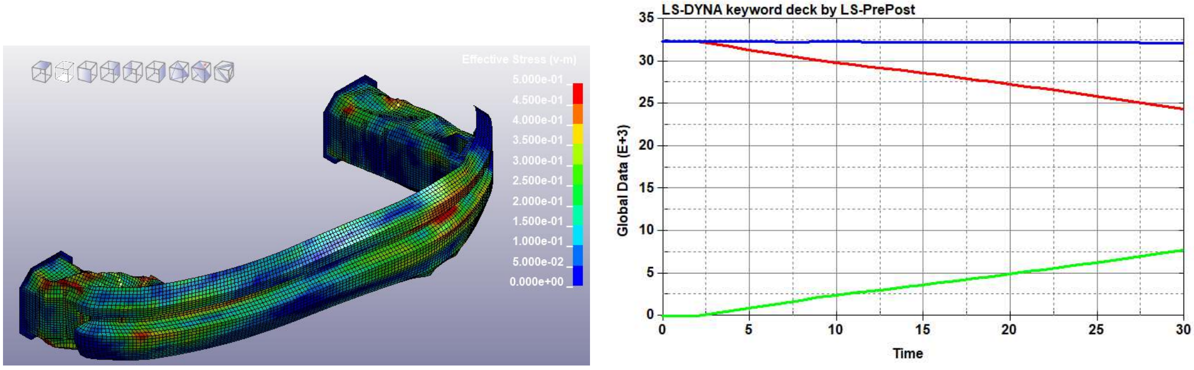
The impact load on the rigid wall can be checked by plotting the data by using the **ASCII** option. In addition, if the timeline is selected from the bottom menu of the plot, the frame showing the deformation of the fringe plot and the frame of the graph can be synchronized and analyzed.

- **ASCII → RWFORC | Load → Wall1 | Plot after selecting normal force**

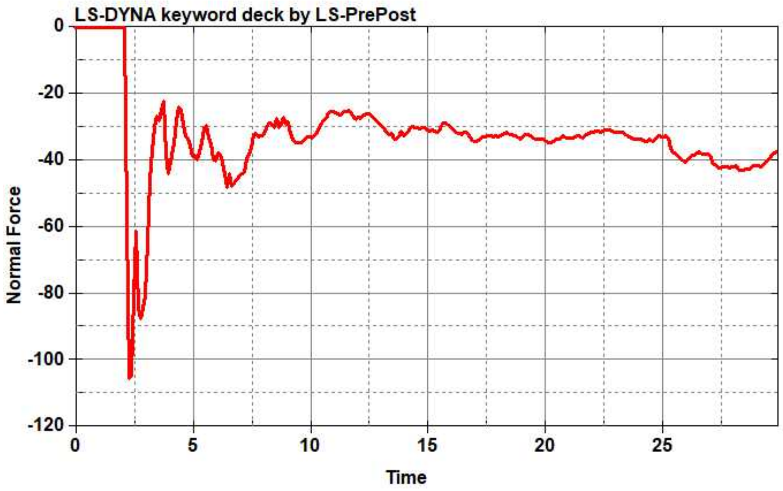


▲ Load and RCFORC Distribution due to Rigid Wall Impact

Create a model for the side collision and then analyze and compare the data with a normal impact. The figure below shows the analysis results and energy distribution of the side impact model. Compare the data below with forward collision data and think about the differences that are occurring.



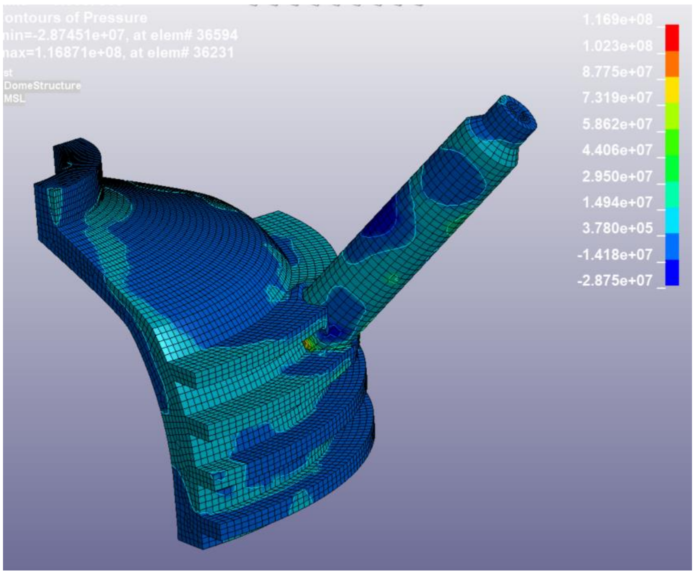
▲ Von mises stress distribution and energy distribution in side-impact conditions



▲ Load and RCFORC distribution due to rigid wall impact (front side collision condition)

So far, we have analyzed the impact-absorbing behavior of the automobile crush box for two collision conditions. Currently, the octagonal crush box shape and the bead initiator design are known to be superior to other shapes with different effects on shock absorption. Using this exercise model to design a more optimized system would be a good learning experience. Recently, in order to obtain a better shock absorbing effect while maintaining a lightweight, it is often manufactured by filling a cellular foam such as urethane foam or sponge inside.

Chapter 9. Penetration Analysis of Missile Projectile



Objectives

- ✓ **Understanding the Concrete Property Model**
- ✓ **Modeling Element Erosion Due to Penetration**
- ✓ **Setting up two-way contact and self contact**

Duration: 60min

Application: Defense, Explicit, Penetration

Model files

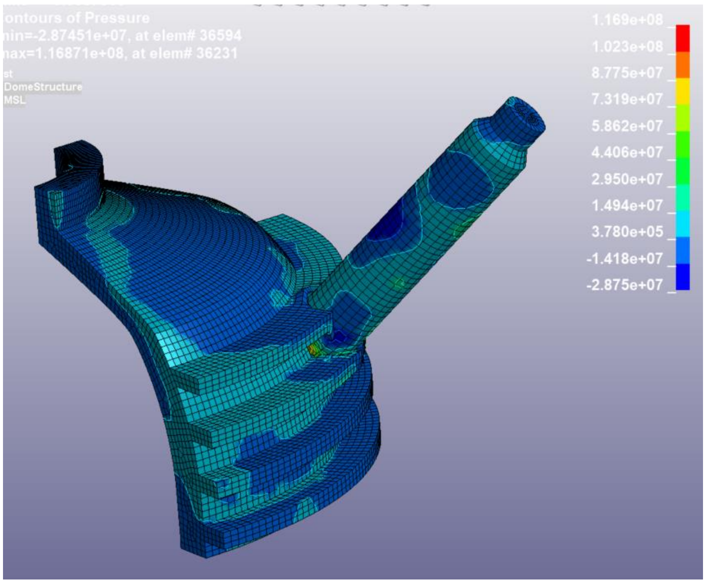
- W9_Initial_Mesh.k: Initial state model (Mesh model)
- W9_SETUP_MSLProjectile.k: Final state model

W9

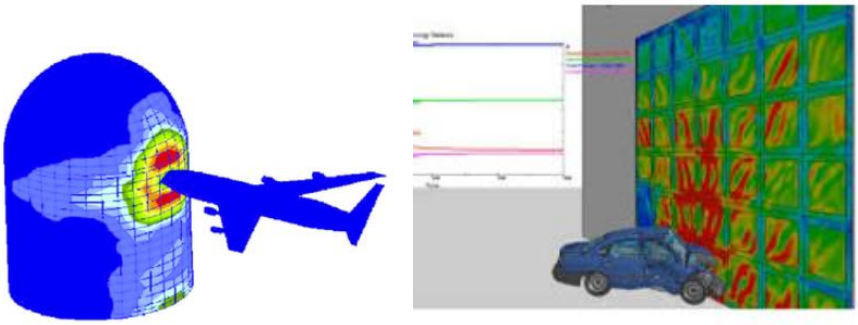
Workshop 9.
Projectile Penetration Analysis of Concrete Wall

Introduction

The design of a protection structure against high-speed impact shocks is one of the most important factors for various security facilities. In this analysis, the most widely used protection structure made up of concrete material which is applied to a condition in which the missile type projectile collides at a speed of several hundred km/h. The design of such a protective structure for key facilities and important persons has received much attention since the 9/11 terrorist attacks. In particular, penetration analysis of concrete structures caused by aircraft or automobiles can be found in recent many papers.



▲ High-Speed Impact Analysis of Missiles on Concrete Structures



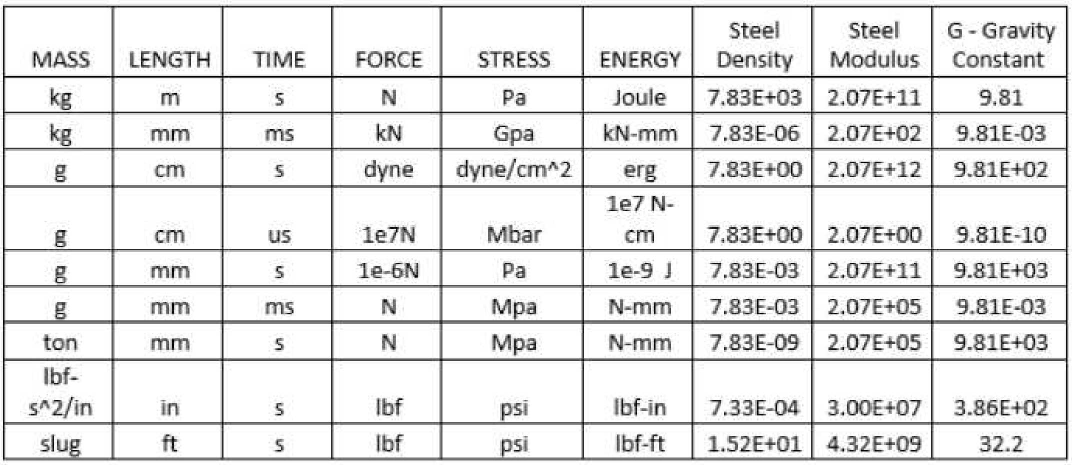
▲ Impact Analysis of Concrete Structures (Cases)

Import Mesh Model

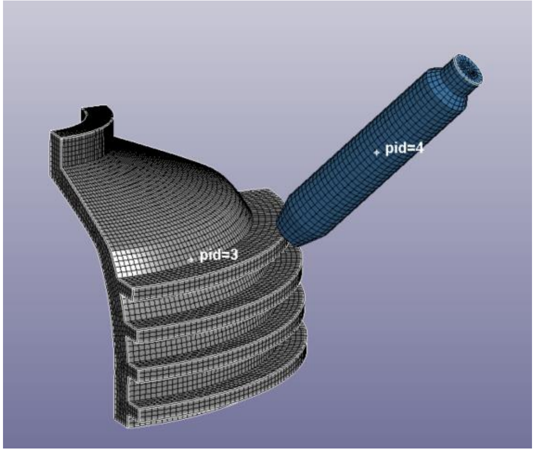
If you open the provided DYNA model file ***'W9_Initial_Mesh.k'*** in the Prepost, the dome-shaped structure and the missile-type projectile will appear in the Prepost as shown below. The unit system is MKS and is as follows.

- **Unit system [m, kg, sec, N, Pa]**

NOTE Since DYNA uses a wide variety of unit systems depending on the model, it is necessary to refer to the chart below to set up the proper unit system. It is not easy to check which unit system with an analysis model performed by another department. Most models of DYNA include density, so we can infer the units of this density. For steel materials, if the density is entered as 7830, it means the meter unit is likely to be used and if it is entered as 7.83e-3, the unit of mm is likely to be used. In this case, additional units such as modulus or velocity should be checked. Please refer to the Appendix for a detailed description of the Unit System.



MASS	LENGTH	TIME	FORCE	STRESS	ENERGY	Steel Density	Steel Modulus	G - Gravity Constant
kg	m	s	N	Pa	Joule	7.83E+03	2.07E+11	9.81
kg	mm	ms	kN	Gpa	kN-mm	7.83E-06	2.07E+02	9.81E-03
g	cm	s	dyne	dyne/cm^2	erg	7.83E+00	2.07E+12	9.81E+02
g	cm	us	1e7N	Mbar	1e7 N-cm	7.83E+00	2.07E+00	9.81E-10
g	mm	s	1e-6N	Pa	1e-9 J	7.83E-03	2.07E+11	9.81E+03
g	mm	ms	N	Mpa	N-mm	7.83E-03	2.07E+05	9.81E-03
ton	mm	s	N	Mpa	N-mm	7.83E-09	2.07E+05	9.81E+03
lbf-s^2/in	in	s	lbf	psi	lbf-in	7.33E-04	3.00E+07	3.86E+02
slug	ft	s	lbf	psi	lbf-ft	1.52E+01	4.32E+09	32.2



▲ Imported Dome Structure and Missile Projectile Mesh Models

Let's take a look at the models imported as parts 3 and 4. Reinforcement structures are designed in the circumferential direction of the ¼ dome-shaped protective structure. The high-speed impact body is shaped like a typical missile that is hollow inside. The element uses a typical Hexa element and the total number of elements uses 28.5k for protective structure and 1.3k for the missile part. Basic information on parts can be found in Part Data.

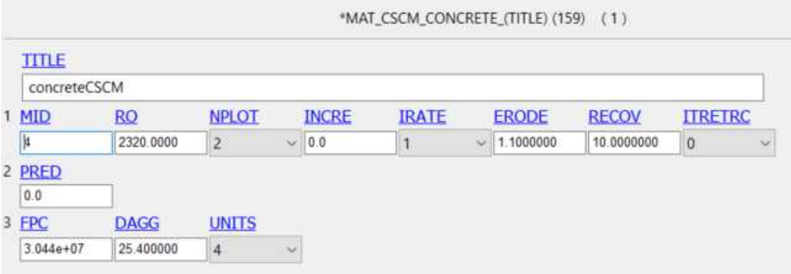
- **Part data → show | select part to measure**

Material Property

Concrete is a major material that appears very often as an analysis subject. Therefore, it is very important to understand the property model for concrete materials. LS-DYNA has three representative concrete material models. These concrete models will be discussed in detail later in the coming pages.

The material model for concrete used in this analysis is **MAT159_MAT_CSCM_Concrete**. The **MAT159** concrete model was developed in the 1990s and has been applied to many studies for a while and has been applied to LS-DYNA since 2005. This material model consists of relatively simple equations and has the advantage of effectively modeling the damage of concrete with Continuous Surface Cap Material (**CSCM**). However, the effect on the strain rate has a disadvantage that a relatively similar result can be obtained only through the calibration procedure. Refer to the figure below for the input values of the **MAT159** property model. **MAT000_ADD_EROSION** is modeled together to apply the function of element erosion of concrete material, that is, the option is evoked to remove the element from the model so that it no longer maintains rigidity.

- **Keyword Manager → MAT | 024-Piecewise_linear_plasticity → input properties**



*MAT_CSCM_CONCRETE_(TITLE) (159) (1)

TITLE: concreteCSCM

	MID	RO	NPLOT	INCRE	IRATE	ERODE	RECOV	ITRETRC
1	4	2320.0000	2	0.0	1	1.1000000	10.0000000	0

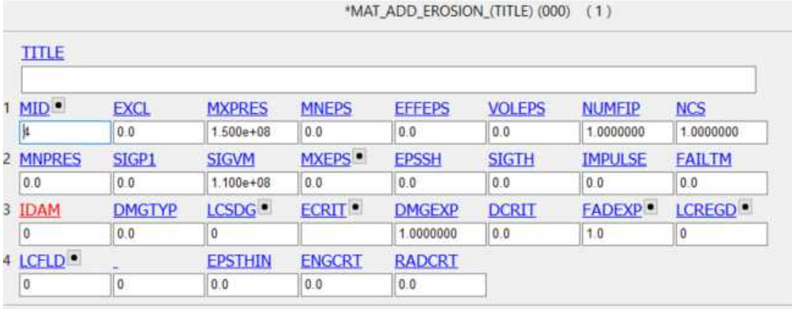
	PRED
2	0.0

	FPC	DAGG	UNITS
3	3.044e+07	25.400000	4

▲ MAT159 Material Properties

NOTE The unit system applied in this workshop tutorial is the meter system. You must select 4 for UNITS on the last line. RO has a density of 2320 kg / m^3 and NPLOT = 2 implies both brittle and ductile damage. If ERODE is less than 1, no element removal will be applied. If the damage is independent of strain, enter 1. FPC is the compressive strength of 30.44 MPa, DAGG is the size of particles in the concrete structure.

MAT_ADD_EROSION is a property keyword that removes an element from the calculation when it exceeds the failure criteria as the analysis proceeds. Enter the ID of the property model (**MAT159** concrete) to apply to MID. Two failure criteria **(MXPRES, SIGVM)** were applied to physical properties. Enter the maximum pressure at the break as 150 MPa and the von Mises stress of 110 MPa as shown below. **NCS** = 1 means erosion occurs if only one criterion is met. If 2 is entered, breakage occurs when both failure conditions are satisfied. In addition, various stress-based and strain-based criteria can be applied.



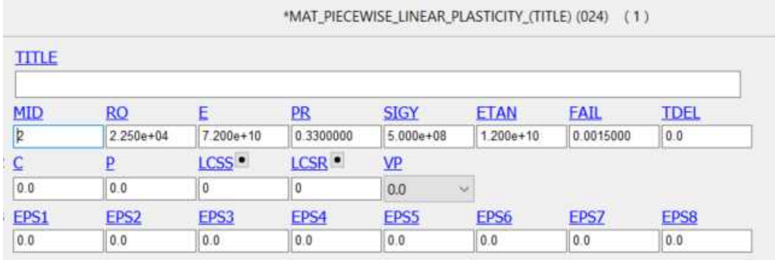
*MAT_ADD_EROSION_(TITLE) (000) (1)

TITLE

1	MID ●	EXCL	MXPRES	MNEPS	EFFEPS	VOLEPS	NUMFIP	NCS
	4	0.0	1.500e+08	0.0	0.0	0.0	1.0000000	1.0000000
2	MNPRES	SIGP1	SIGVM	MXEPS ●	EPSSH	SIGTH	IMPULSE	FAILTM
	0.0	0.0	1.100e+08	0.0	0.0	0.0	0.0	0.0
3	IDAM	DMGTYP	LCSDG ●	ECRIT ●	DMGEXP	DCRIT	FADEXP ●	LCREGD ●
	0	0.0	0		1.0000000	0.0	1.0	0
4	LCFLD ●	_	EPSTHIN	ENGCRT	RADCRT			
	0	0	0.0	0.0	0.0			

▲ MAT000_ADD_EROSION setup

Next, enter the commonly used **MAT24** properties to the missile's metal exterior, referring to the figure below. Physical material properties were set based on aluminum. The density was largely set in consideration of various warheads and propellants inside the missile. In this analysis, plastic deformation does not need to be entered because deformation in the plastic zone is not considered. Breakage criteria were set to occur when strain exceeded 0.0015.



*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE

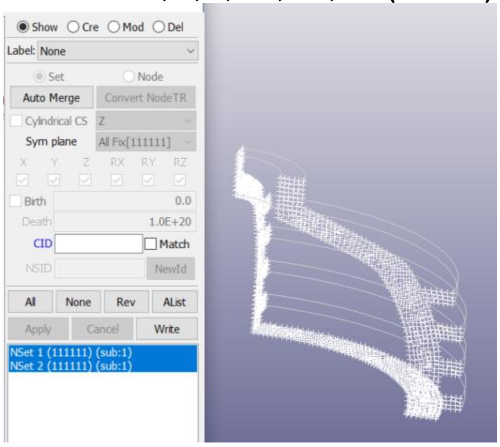
MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
2	2.250e+04	7.200e+10	0.3300000	5.000e+08	1.200e+10	0.0015000	0.0
C	P	LCSS ●	LCSR ●	VP			
0.0	0.0	0	0	0.0			
EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

▲ MAT24_PIECEWISE_LINEAR_PLASTICITY setup

Boundary Condition

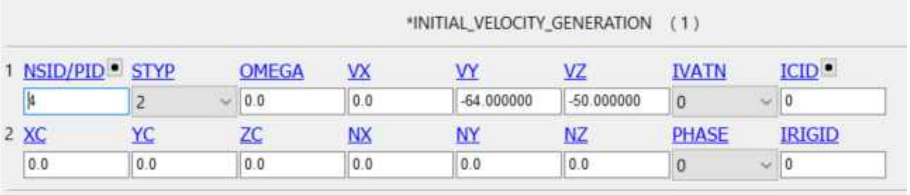
Fixation is applied on both the bottom and side of the Dome structure. The high-speed impact body was velocity is set to impact at a slope of about 40 degrees at VY = -64 m/s and VZ = -50 m/s.

- **Create Entity → SPC | Cre → Select | Pick | By element | Prop, angle=5 → select Dome bottom and side surfaces → check X, Y, Z, RX, RY, RZ (111111) → Done**


● Show ○ Cre ○ Mod ○ Del
Label: None
● Set ○ Node
☐ Cylindrical CS: Z
Sym plane: All Fix[111111]
X ☑ Y ☑ Z ☑ RX ☑ RY ☑ RZ ☑
☐ Birth: 0.0
Death: 1.0E+20
CID: ☐ Match
NSID
NSet 1 (111111) (sub:1)
NSet 2 (111111) (sub:1)

▲ Boundary condition, SPC setup

- **Keyword Manager → Initial | Velocity Generation → see below image, Enter the part ID as the missile part id**


*INITIAL_VELOCITY_GENERATION (1)

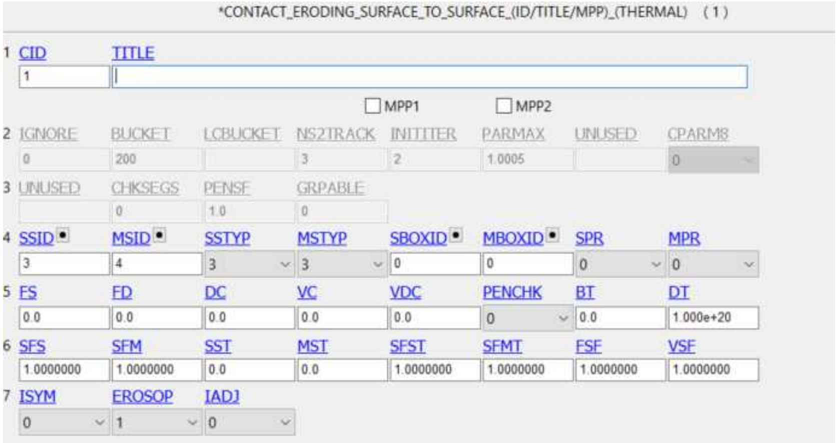
1	NSID/PID	STYP	OMEGA	VX	VY	VZ	IVATN	ICID
	4	2	0.0	0.0	-64.000000	-50.000000	0	0
2	XC	YC	ZC	NX	NY	NZ	PHASE	IRIGID
	0.0	0.0	0.0	0.0	0.0	0.0	0	0

▲ Initial Velocity setup

Contact Condition

For the contact condition of the dome structure and missile, apply ***Contact_eroding_surface_to_surface** as shown below to invoke the element's bidirectional erosion. Master and Slave surfaces are set by Part ID and missiles are set to Master and protective structures are set to Slave. Note that 1 must be entered in **EROSOP**. Note that if you do not enter 1, element erosion will occur only at the exterior surface of the initial impact and no contact at the internal elements will occur.

- **Keyword Manager → Contact | *Contact_eroding_surface_to_surface → see image**

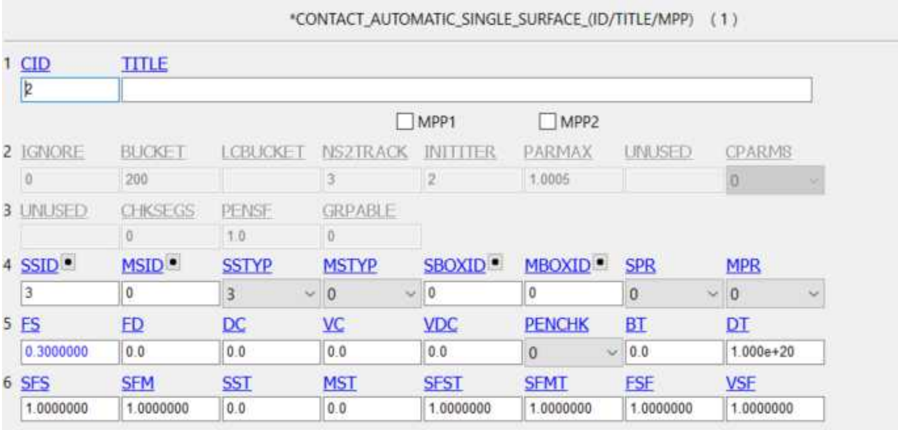


*CONTACT_ERODING_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (1)

1	CID	TITLE						
	1							
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ■	MSID ■	SSTYP	MSTYP	SBOXID ■	MBOXID ■	SPR	MPR
	3	4	3	3	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.0	0.0	0.0	0.0	0.0	0	0.0	1.000e+20
6	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
	1.0000000	1.0000000	0.0	0.0	1.0000000	1.0000000	1.0000000	1.0000000
7	ISYM	EROSOP	IADJ					
	0	1	0					

▲ Surface contact setup

In case of structural damage due to such impact, self-contact may occur, so set the self-contact condition on the dome structure or missile structure. For applying self-contact, use the * **Contact_automatic_single_surface** keyword. Although it has a bidirectional contact type, if all master surfaces are set to 0, self-contact occurs only on the slave surface, so set the dome structure (SSID = 3) as a slave as shown below.



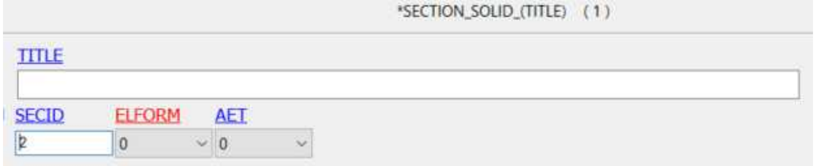
*CONTACT_AUTOMATIC_SINGLE_SURFACE_(ID/TITLE/MPP) (1)

1	CID	TITLE						
	2							
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ■	MSID ■	SSTYP	MSTYP	SBOXID ■	MBOXID ■	SPR	MPR
	3	0	3	0	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.3000000	0.0	0.0	0.0	0.0	0	0.0	1.000e+20
6	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
	1.0000000	1.0000000	0.0	0.0	1.0000000	1.0000000	1.0000000	1.0000000

▲ Self contact condition setup

Section and Part Setup

In the model consisting of solid elements, all sections are of * **Section_Solid** and **Elform = 0** is applied. The section is set up like this and input the Part property of the model.



*SECTION_SOLID_(TITLE) (1)

TITLE		
SECID	ELFORM	AET
2	0	0

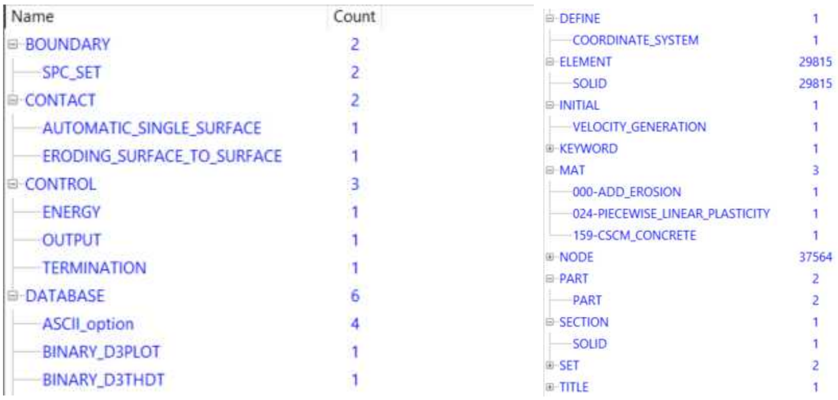
- **Keyword Manager → Part → Enter Material ID and Section ID for Missile and Dome Structure**

Analysis step

The analysis is set up to generate 40 D3PLOTs in about 0.01 sec.

- **Keyword Manager → Control | Termination → ENDTIM = 0.01**
- **→ ASCII_option → Default DT=0.002, select GLSTAT, MATSUM, NODOUT, RWFORC**
- **→ BINARY_D3PLOT → DT=40**
- **→ BINARY_D3THDT → DT=40**
- **→ KEYWORD → MEMORY=800000000, NCPU=4 (if your PC has 4 CPU)**
- **→ ENERGY → HGEN=2, RWEN=2, SLNTEN=2, RYLEN=1**
- **→ KEYWORD → MEMORY=800000000, NCPU=4 (if your PC has 4 CPU)**

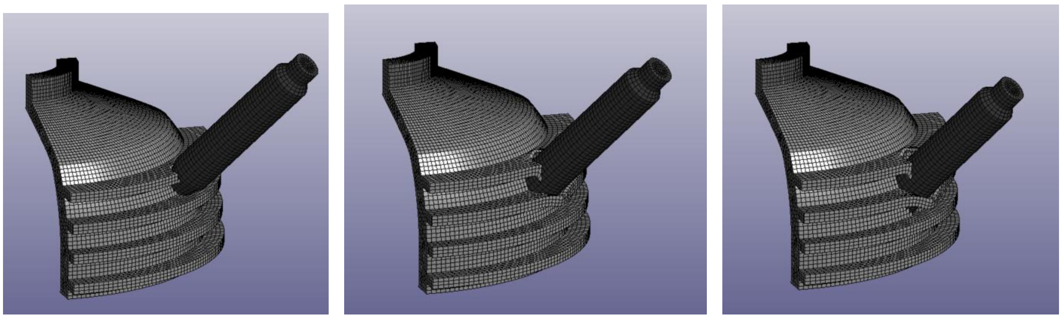
If you have completed this step, the keyword generation for modeling is complete. Let's open the **Keyword Manager** and check the generated keywords. The following keyword should have been created:



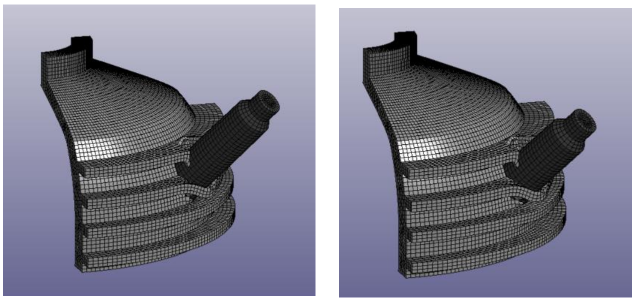
Name	Count
BOUNDARY	2
SPC_SET	2
CONTACT	2
AUTOMATIC_SINGLE_SURFACE	1
ERODING_SURFACE_TO_SURFACE	1
CONTROL	3
ENERGY	1
OUTPUT	1
TERMINATION	1
DATABASE	6
ASCII_option	4
BINARY_D3PLOT	1
BINARY_D3THDT	1
DEFINE	1
COORDINATE_SYSTEM	1
ELEMENT	29815
SOLID	29815
INITIAL	1
VELOCITY_GENERATION	1
KEYWORD	1
MAT	3
000-ADD_EROSION	1
024-PIECEWISE_LINEAR_PLASTICITY	1
159-CSCM_CONCRETE	1
NODE	37564
PART	2
PART	2
SECTION	1
SOLID	1
SET	2
TITLE	1

▲ Generated keyword

This is the final stage of FEM modeling. Compare the generated keyword with the provided keyword ‘***W9_SETUP_MSLProjectile.k’*** to check the differences, make corrections and save your .k file. Run the analysis and load the result file **D3PLOT** from the Prepost. A d3plot for 40 frames has been created and you will see the collision as shown below.



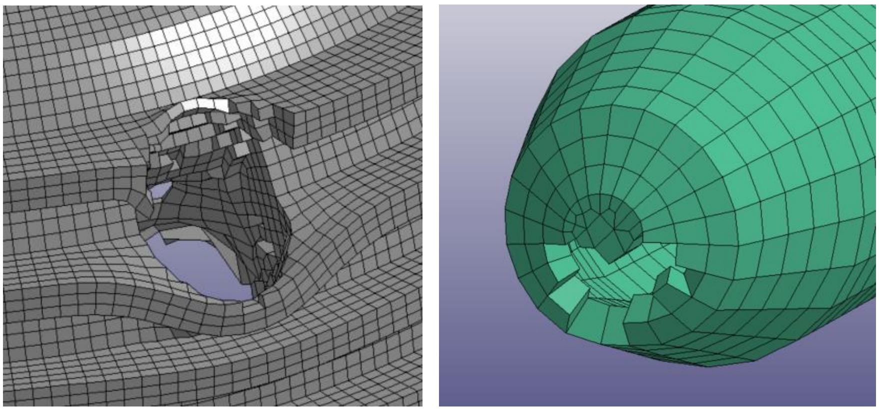
0.002 sec 0.004 sec 0.006 sec



0.008 sec 0.010 sec

▲ Impact process

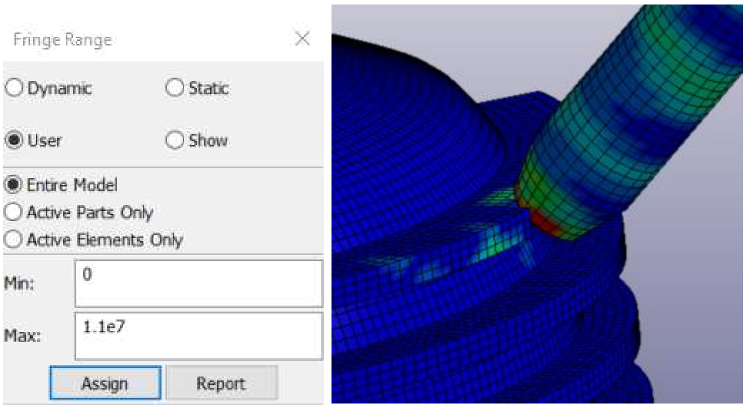
You can see how the concrete barrier is penetrated as the element is removed due to the missile collision. This phenomenon can be visualized because element erosion was included in the keyword model. Without element erosion, abnormal deformations of the element could be observed, or the results could be in the form of a missile penetrating without imparting any damage. The figure below shows the parts by which the element was removed beyond the damage criteria.



▲ Element erosion site

Use the post-processing function to check the stress level in the region of the collision area. If you check the stress level by setting the Fringe range area as the damage criterion Max = 110MPa, you can see that the stress level of the concrete wall exceeds 110MPa from the beginning of the impact as shown below. It is the time when the element failure occurs with the collision.

- **Post → Fringe component | Stress | Pressure or Von mises stress**
- **Fringe range → User | Entire Model | Min=0, Max=1.1e7**



Fringe Range

○ Dynamic ○ Static

● User ○ Show

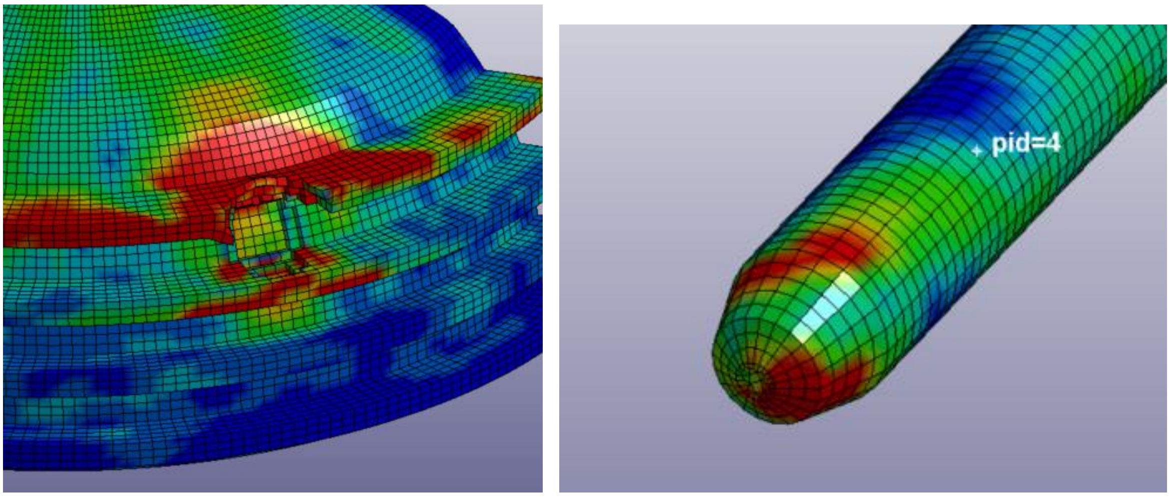
● Entire Model
○ Active Parts Only
○ Active Elements Only

Min: 0

Max: 1.1e7

▲ Pressure distribution

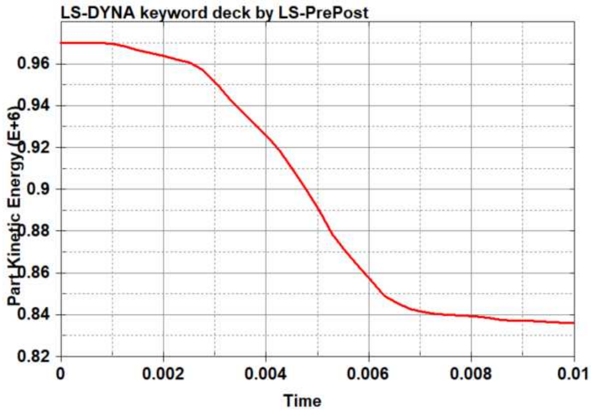
The figure below shows the von Mises stress distribution by component. It can be seen that during the initial impact, the dome circumferential reinforcement structure supports much of the stress.



▲ Von Mises stress distribution

Let's check the kinetic energy of the missile part in history. As shown in the figure below, the kinetic energy shows a tendency to decrease rapidly during the penetrating process after the initial collision and then to be maintained.

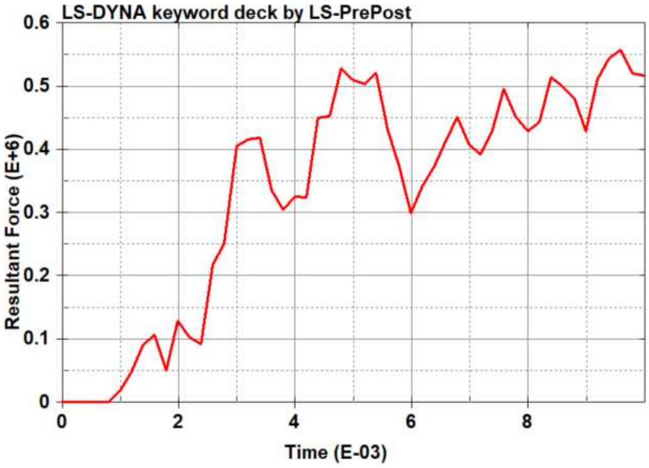
- **History → Part | Sel. Part, select missile, by part | Kinetic energy → Plot**



▲ Kinetic energy distribution of missiles

The impact load on the concrete wall can be checked by plotting the data using the ASCII option. In addition to this, let's compare the strain, total energy, and stress distribution at the node of a specific part with various collision processes.

- **ASCII → RWFORC | Load → sl-1 | Resultant force → plot**



▲ Impact load RCFORC distribution on the barrier

Knowledge

Understanding of Concrete Property Model

Concrete is one of the most widely used materials for modern structures, buildings, and infrastructure. LS-DYNA offers a variety of concrete models. The analysis of structural materials and facilities using concrete is also a common task in industrial fields or in schools. Therefore, a general explanation will be given to those who have a lack of understanding of concrete structures. The DYNA version covered in this book is R-970, so if there is a difference between the textbook, please check your installed version. The characteristics to be dealt with in general concrete models include elastic behavior, softening, shear dilation and strain rate effect. There are three representative concrete models. Most concrete structures are in the form of Reinforced Concrete (RC) with steel beams inside. The proper model should be established by accurately identifying these characteristics and whether the environment to be analyzed is quasi-static, blast, or impact loads. Using a validated model to predict realistic dynamic behavior is an important factor for both industrial engineers and academic activities in schools.

- **MAT072 (KCC) MAT_Concrete_Damage**
- **MAT084 (Winfrith) Mat_Winfrith_Concrete**
- **MAT159 (CSCM) Mat_CSCM_Concrete**

In addition, there are many different models, each with its own particular features. The typical models listed above have been validated by various experiments and studies and another important advantage is that they are relatively simple. The material characteristics of concrete are quite different from those of metals. Concrete is strongly influenced by its behavior depending on the pressure applied. Yield strength (Yield, Y) of the material is expressed by the following formula. I_1 means 1st invariant of stress tensor and J_2 and J_3 mean second and third deviatoric stress tensors, respectively. For a detailed explanation of the concepts used in solid mechanics, please check the solid mechanics textbook if necessary.

$$\mathrm{Y}(I_1, J_2, J_3) = 0$$

In general, various models of concrete are classified according to how they simulate deviatoric or volumetric behavior.

- **MAT072 (KCC) MAT_Concrete_Damage**

MAT072, also known as the **KCC** model, is an abbreviation of Karagozianr & Case Concrete. It was developed for an explosion or impact analysis of RC concrete and was applied to DYNA in 2004. Just enter a few simple compressive strengths and densities to compute similar parameters using your own algorithm. Readers who wish to use in academic research should refer to the manual on the verification of physical property models. The KCC model is constructed with three independent strength planes. i = y, m, r are expressed as yield strength surface, maximum strength surface and residual strength surface, respectively. P = -I / 3 means pressure. These variables are obtained through tests, and the governing equations of the property model are expressed based on these test values.

$$F_i(p) = a_{0i} + \frac{p}{a_{1i} + a_{2i}p}$$

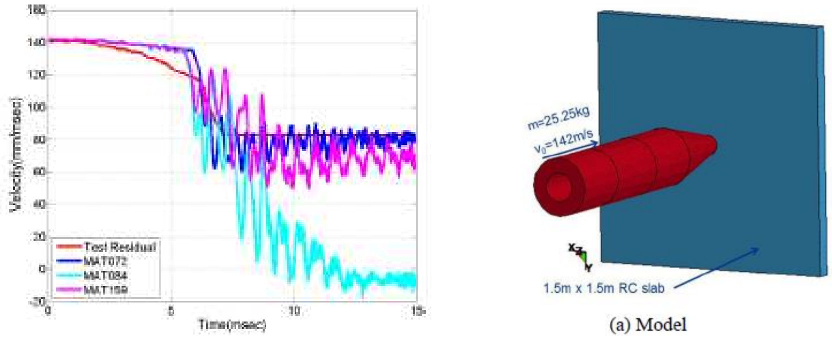
More complex theoretical explanations are beyond the scope of this book, so I hope you will gain a theoretical background through individual learning. Please understand that this model is suitable for impact or explosion analysis of general concrete structures.

- **MAT084 (Winfrith) Mat_Winfrith_Concrete**

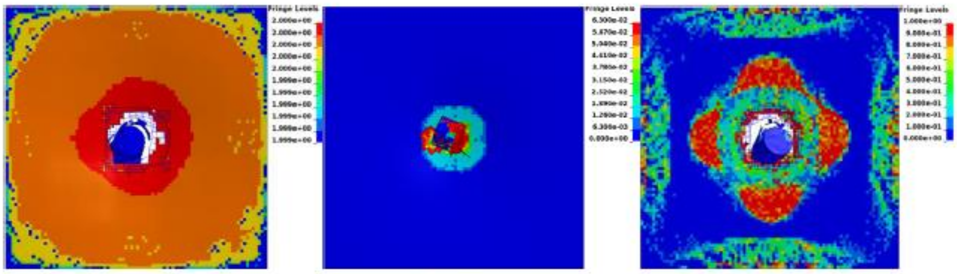
Winfirith's concrete model was developed in the 1980s and included in DYNA in 1991, which is widely used for general analysis. Although not as simple as the KCC model, it is a model that can easily input complex concrete material properties. Another advantage is that crack analysis can be performed in three directions per element and the analysis results can be confirmed in the prepost. It was mainly developed for the analysis of crack initiation during impact analysis of RC concrete. The governing equation is composed of four variables as below and the variables can be calculated by correlation of tensile strength and compressive strength.

$$\Upsilon(I_1, J_2, J_3) = aJ_2 + \lambda\sqrt{J_2} + bI_1 - 1$$

The figure^{A} below shows the differences and characteristics of each model as a result of analyzing the impact of three concrete models on the same projectile.



(a) Model



(a) MAT072 (b) MAT084 (c) MAT159

- **MAT159 (CSCM) Mat_CSCM_Concrete**

The most recently developed model, which was applied in this analysis, was included in DYNA 2005. As an update to the **KCC** model, this model automatically calculates related factors with a few simple parameters. Thus, MAT159 is one of the widely applied models for a concrete structure. The verification of the model can be found in the reference paper. The governing equation is as follows and the yielding behavior is defined by three stress invariants. Readers who want to understand theoretically should check the reference paper.

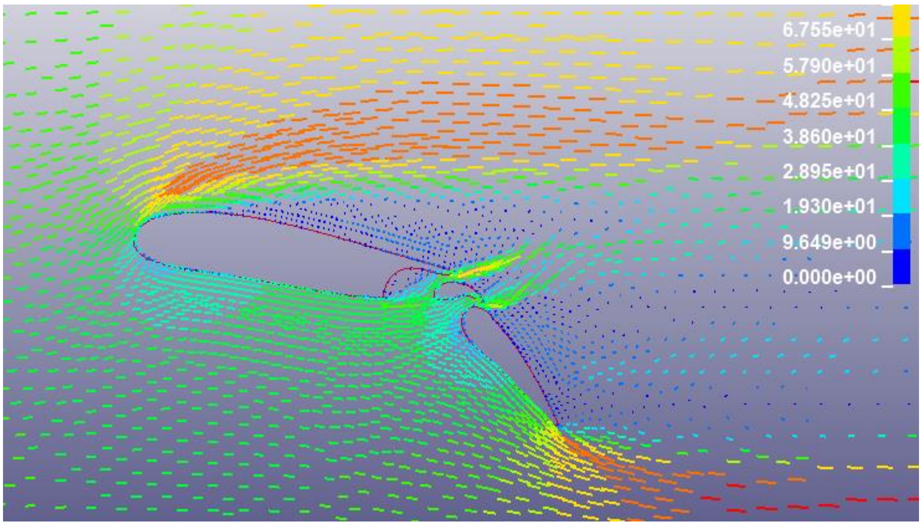
$$Y(I_1, J_2, J_3) = J_2 - \Re(J_3)^2 F_f^2(I_1) F_c(I_1, \kappa)$$

The three concrete properties models have been briefly examined and the properties of the **MAT159** model applied in this analysis are attached.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$$$$  MAT159 CSM Concrete Properties
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
 *MAT_CSCM_CONCRETE_TITLE
Concrete CSCM
$#    mid       ro    nplot    incre    irate    erode    recov  itretrc
        4   2320.0        2      0.0        1      1.1     10.0        0
$#   pred
      0.0
$#    fpc     dagg    units
3.044000E7  0.0254        4
```

[A] Y. Wu, J E. Crawford, J M. Magallas “Performance of LS-DYNA Concrete Constitutive Models”, 12th International LS-DYNA User Conference

Chapter 10. 2D Wing CFD Analysis with ICFD Solver



Objectives

- ✓ Learn how to use ICFD Solver
- ✓ Practice how to create Volume Mesh
- ✓ Get used to CFD modeling procedure
- ✓ Results visualization of 2D Wing(Flap down) CFD Analysis

Duration: 80min

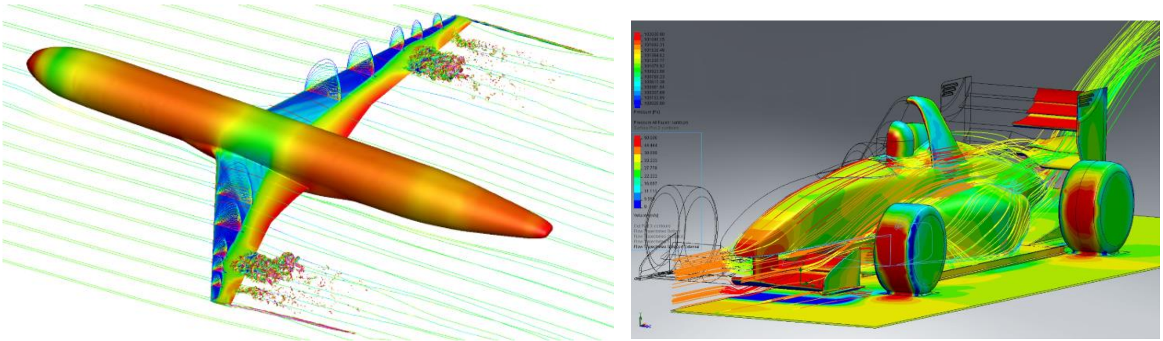
Application: Aerospace, ICFD, 2D Wing Analysis

Model files

- W10_Wing_Profile.k: Initial state model
- W10_INITIAL_Wing_Mesh.k: Initial state model
- W10_SETUP_2Dwing.k: Final state model

Introduction ICFD

Incompressible Computational Fluid Dynamics (ICFD), which has been available since the release of the R7 version of LS-DYNA, is a solver for the simulating incompressible CFD problems. In various industries, it is particularly applicable to the problems of fluids that are related to land vehicles, subsonic aircraft, marine structures, and ships. Although only fluid-related analysis is possible, it has recently been applied to solve a fluid-structure interaction (FSI) problem, which is an interlocking analysis with a structure. Previously, according to the FEM classification of CFD and structural analysis were different and even the Engineering graduate departments and various curriculums were organized based on this. However, in recent industrial fields, multiphysics analysis that combines both fields, including thermal mechanics, is widely applied. Therefore, to be a competent engineer, you need to understand various fields.



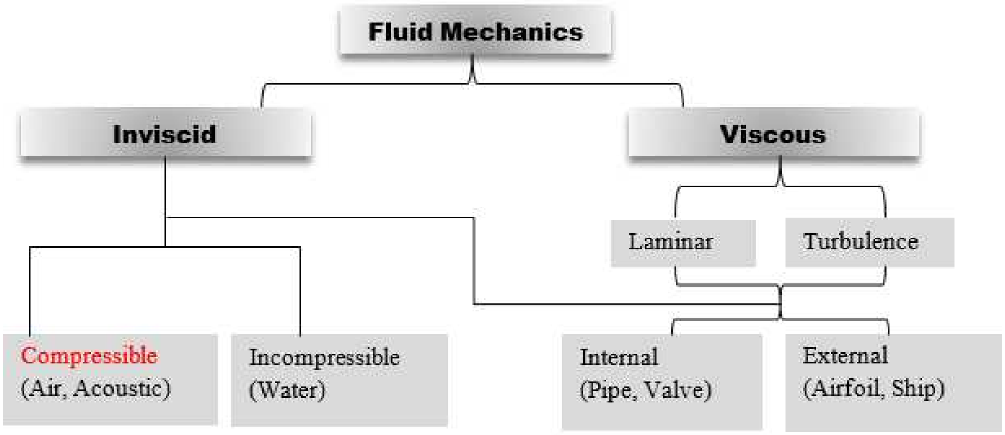
▲ Various CFD Analysis Case[7]

Basically, the ICFD solver of LS-DYNA does not provide many functions compared to other CFD dedicated analysis programs. The ICFD solver is a FEM based implicit solver for incompressible fluids. Most popular CFD analysis programs use the finite difference method (FDM) or finite volume method (FVM).

Basically, the fluids fields covered in the CFD domain are as follows. The area which is not provided by LS-DYNA is the Compressible flow area and the basic turbulence model is provided in the LS-DYNA. In addition, it provides Volume Mesh function, which is the core function in the CFD field and provides interworking analysis functions with other solvers. Another major function is the boundary layer mesh function. It can be very useful in the analysis of the behavior where the shear stress on the surface is important. This feature will be used in this practice model.

---

[7] Image courtesy of ANSYS conference proceedings



▲ CFD categorization

One of the main uses of ICFD provided by LS-DYNA is not only to solve the Navier Stokes application but also to solve the problem of a fully coupled fluid-structure interaction with the Lagrangian model. Both implicit and explicit solvers in the field of structural analysis can be integrated with ICFD. The ability to send and receive stresses and displacements to each other in one direction or in both directions is provided.

One of the main parameters that define the characteristics of a fluid is the Reynolds Number. The Reynolds number can be calculated by the formula from the density of the fluid, the size of the structural material and the size of the dynamic viscosity. In general, it is known that turbulence starts to form from 150 and above and complete turbulence occurs at 300 or more.

$$\mathrm{Re} = \frac{\rho v l}{\mu} = \frac{v l}{\nu}$$

Where:

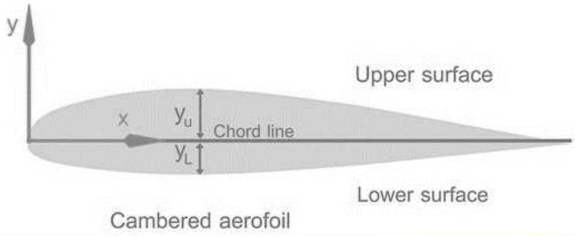
v = Velocity of the fluid
l = The characteritics length, the chord width of an airfoil
ρ = The density of the fluid
μ = The dynamic viscosity of the fluid
ν = The kinematic viscosity of the fluid

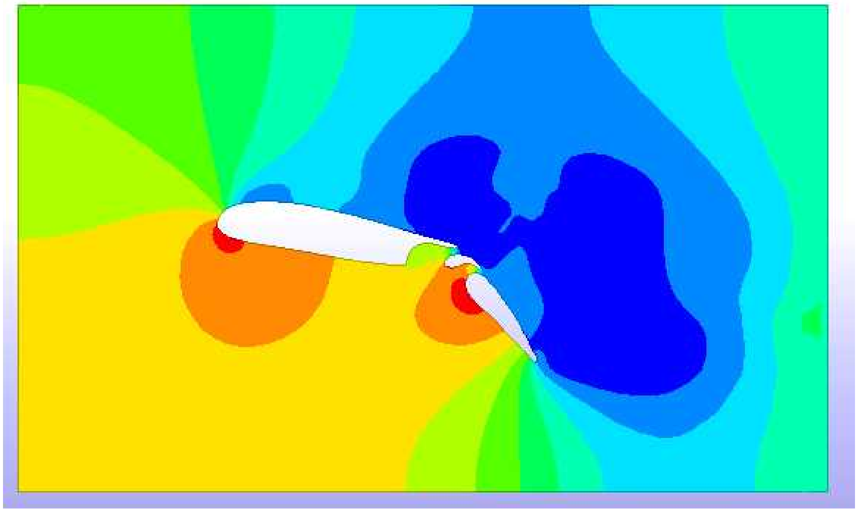
Selecting an appropriate turbulence model in CFD analysis is a very important factor in modeling the vortices occurring in the flow. Representative RANS and LES models are available from DYNA. In addition, the ICFD solver enables heat transfer analysis and non-Newtonian fluid analysis so that the basic fluid analysis functions can be solved with one DYNA solver. It can be said to provide enough functions as a multiphysics analysis program.

W10 Workshop 10.
2D Wing Airfoil (Flap down) CFD Analysis

Introduction

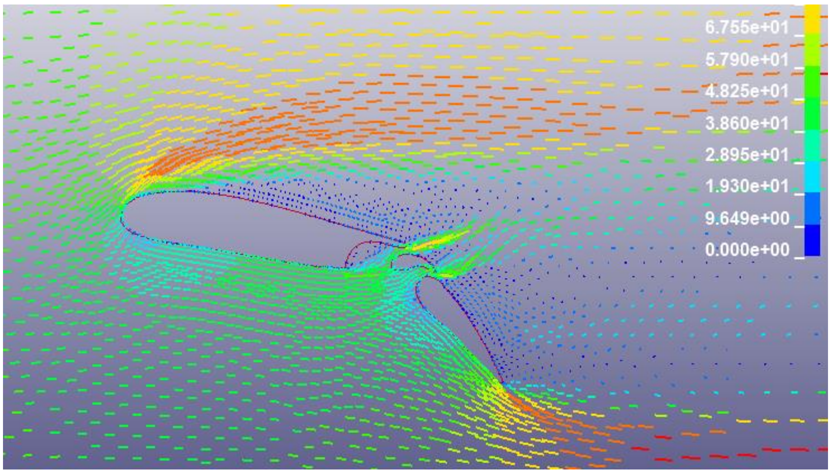
The most important structure in an aircraft is the wings and the cross-section of the wing is called Airfoil, which is the key mechanism to help generate lift and helps the aircraft to fly. In this exercise, we will analyze the airflow of a 2D airfoil with a flap by using an ICFD solver. We will proceed fluid analysis at 50m/s speed after down the flap attached to the wing with a 3.5m wing cord. Basically, because the Reynolds number is very large, turbulence is formed and the condition in the analysis is to raise the angle of attack for landing by lifting the flap or lowering the flap.





▲ Pressure distribution analysis result of 2D wing airfoil with flap down

The following figure shows the result of plotting the analysis result of the velocity as a vector. With a high angle of attack and a flap down, the air is flowing at a rate of 50 m/s. Therefore, it can be seen that since the separation of the airflow occurs after the upper surface-specific point, turbulence is formed. The 2D Wing airfoil analysis itself using ICFD is not very complicated. Although ICFD solver provides a good volume mesh creation, user convenience is a little lower, so if there is a preprocessor that is used mainly by a user, using it is a good way for achieving effective modeling.



▲ Velocity distribution result of 2D wing airfoil with flap down

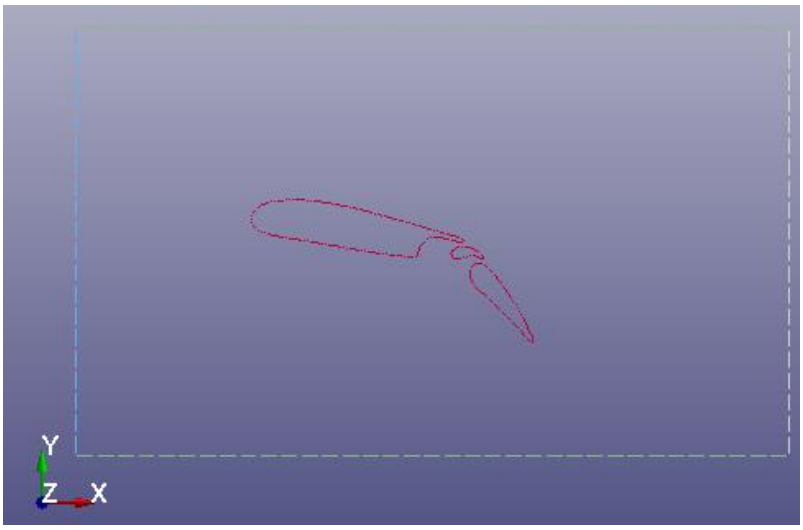
The 2D Wing Airfoil modeling procedure for ICFD analysis can be summarized as follows.

- **Create Geometry**
- **Boundary Element generation → MS Mesh Conversion**
- **ICFD analysis setup (Boundary condition, flow velocity, material/Section assignment)**
- **ICFD Mesh generation (Boundary mesh generation, Volume Mesh generation)**
- **Database Setup**

Import Mesh Model

Import the model ***W10_INITIAL_Wing_Mesh.k*** for flow analysis. Basically, the FEM mesh used in structural analysis and the solver mesh used in ICFD has different characteristics. Therefore, after creating FEM geometry and boundary mesh at the edge/curve, it needs to switch from LS-Prepost to **MS (Multi-Solver) Mesh**. The imported mesh model is a model that has gone through the **MS** process and then generates an ICEM keyword. When the model is imported, the volume mesh is not created as shown in the figure below, but the mesh is created only on the profile lines. The unit of the model is the MKS system.

- **Unit: Meter, sec, kg, N, Pa**

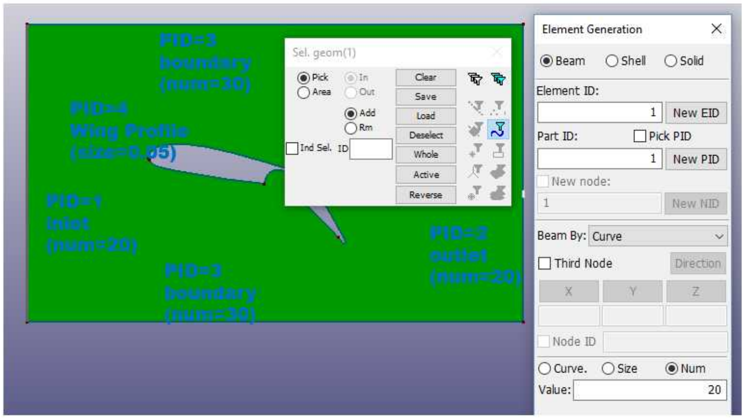


▲ Imported Mesh model

Mesh Generation

First, import the generated geometry model (W10_Wing_Profile.stp). After importing, a model with a wing profile inside the 10m x 6m rectangle will be displayed, as shown in the following figure.

- **File → Import | Step file → select model → Option | meter, 1.0 → Ok**
- **Element and Mesh → Element Generation → Beam, Beam by = Curve, Num=20 → select front Curve, PID=1 → Ok (create element on edge)**

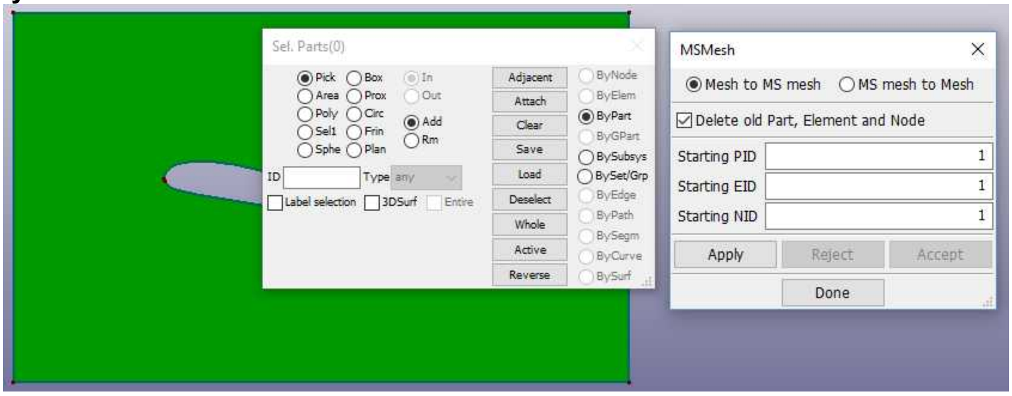


● Beam
Element ID: 1
Part ID: 1
Beam By: Curve
● Num
Value: 20

▲ Beam element generation

If the beam element is created on every curve and Part ID is set, this structural FEM mesh should be converted to **Multi-Solver Mesh which is** recognized by ICFD solver. Save this file as W10_Wing_mesh.k after converting it to **MS mesh**. When you load this file, you will see a model that looks like the previously created mesh. The volume mesh is automatically generated by the ICFD solver which is based on the existing beam elements during the analysis process. The disadvantage is that the generated grid cannot be modified during the modeling process.

- **Elem and Mesh → MSMesh | Mesh to MS mesh, check Delete-, Start ID=1 → select Edge → Apply**



MSMesh

● Mesh to MS mesh ○ MS mesh to Mesh

☑ Delete old Part, Element and Node

Starting PID 1

Starting EID 1

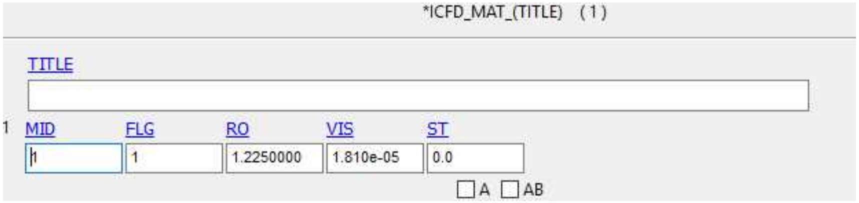
Starting NID 1

▲ Multi-Solver mesh

Material Property

The * **ICFD_MAT** keyword is created to specify the properties of the fluid domain. Enter 1.225 kg/m^3 air density and 1.810e-5 kg/(ms) kinematic viscosity.

- **Keyword Manager → ICFD_MAT → Input material property**



*ICFD_MAT_(TITLE) (1)

TITLE

	MID	FLG	RO	VIS	ST
1	1	1	1.2250000	1.810e-05	0.0

☐ A ☐ AB

Element Property

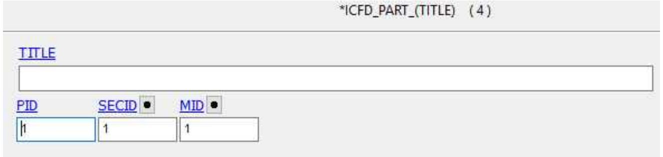
Set the section of the fluid section to * **ICFD_SECTION**. There is no separate input variable.

- **Keyword Manager → ICFD_SECTION → Accept**

Part Property

Set the section and mat to four parts with **MS Mesh** set up (inlet = PID1, outlet = PID2, up / bottom edge = PID3, wing profile = PID4).

- **Keyword Manager → ICFD_Part (1,2,3,4), SECID=1, MAT=1 입력 → Accept**

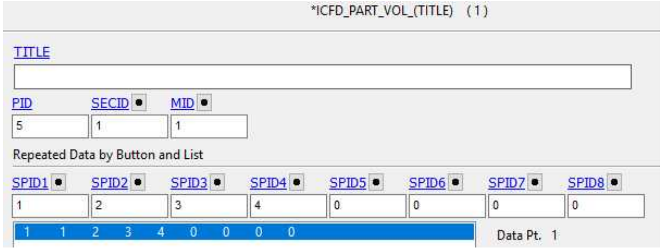

*ICFD_PART_(TITLE) (4)

TITLE

PID: 1
SECID: 1
MID: 1

All four parts are selected through ICFD_Part_Vol to set the volume of the fluid domain.

- **Keyword Manager → ICFD_Part_Vol → select 4 parts, insert →Accept**


*ICFD_PART_VOL_(TITLE) (1)

TITLE

PID: 5
SECID: 1
MID: 1

Repeated Data by Button and List

SPID1	SPID2	SPID3	SPID4	SPID5	SPID6	SPID7	SPID8
1	2	3	4	0	0	0	0

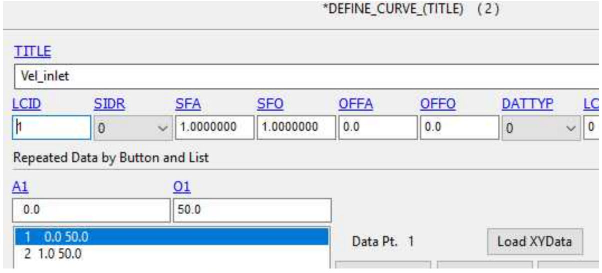
1 1 2 3 4 0 0 0 0

Data Pt. 1

Boundary Condition

Set the boundary condition at the edge of each part as shown below. Velocity and pressure, the basic data for setting boundary conditions, are created through **Define_Curve**. The initial velocity of the fluid entering the inlet is maintained at 50m/s for the analysis time and two curves are generated so that the outlet pressure is continuously zero.

- **Keyword Manager → Define_Curve → Input A1 and Q1 for velocity and pressure (see image)**


*DEFINE_CURVE_(TITLE) (2)

TITLE: Vel_inlet

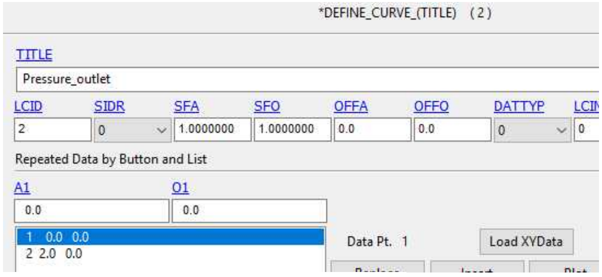
LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	LC
1	0	1.0000000	1.0000000	0.0	0.0	0	0

Repeated Data by Button and List

A1	O1
0.0	50.0

1 0.0 50.0
2 1.0 50.0

Data Pt. 1


*DEFINE_CURVE_(TITLE) (2)

TITLE: Pressure_outlet

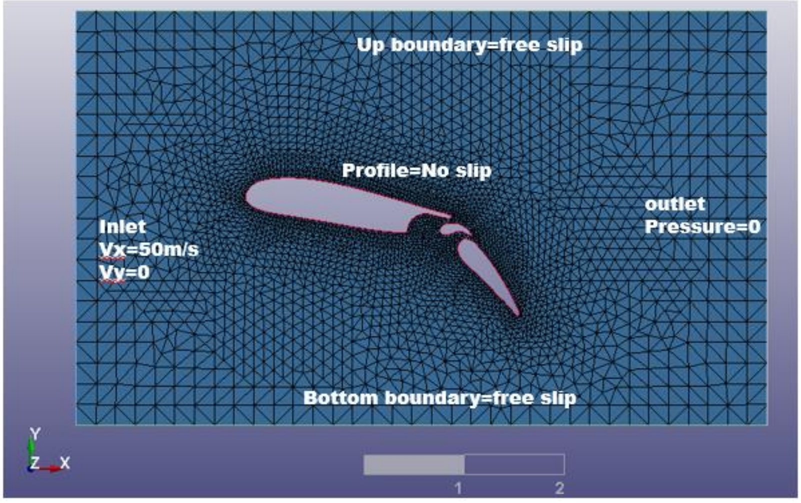
LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	LCI
2	0	1.0000000	1.0000000	0.0	0.0	0	0

Repeated Data by Button and List

A1	O1
0.0	0.0

1 0.0 0.0
2 2.0 0.0

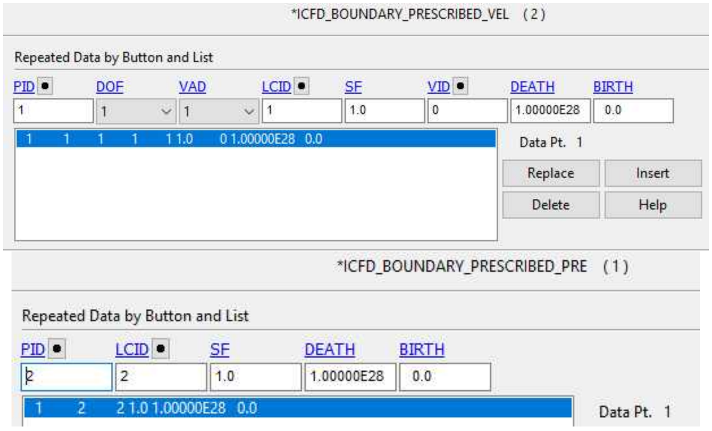
Data Pt. 1



▲ ICFD model boundary condition

The next step is to set the boundary conditions using the defined curves. Set Vx = 50 m / s and Vy = 0 m / s for the inlet part with a pressure = 0 for the outlet part.

- **Keyword Manager → ICFD_Boundary_Prescribed_Vel → PID=1, DOF=1(x axis), VAD=1, LCID= velocity curve → Accept**
- **Keyword Manager → ICFD_Boundary_Prescribed_Vel → PID=1, DOF=2(y axis), VAD=1, LCID= pressure curve → Accept (Vy=0)**
- **Keyword Manager → ICFD_Boundary_Prescribed_Pressure → PID=2, LCID= pressure curve → Accept**



*ICFD_BOUNDARY_PRESCRIBED_VEL (2)

Repeated Data by Button and List

PID	DOF	VAD	LCID	SF	VID	DEATH	BIRTH
1	1	1	1	1.0	0	1.00000E28	0.0

```
1    1    1    1    1 1.0    0 1.00000E28  0.0
```

Data Pt. 1

*ICFD_BOUNDARY_PRESCRIBED_PRE (1)

Repeated Data by Button and List

PID	LCID	SF	DEATH	BIRTH
2	2	1.0	1.00000E28	0.0

```
1    2    2 1.0 1.00000E28  0.0
```

Data Pt. 1

▲ Initial Boundary Condition setup for ICFD model

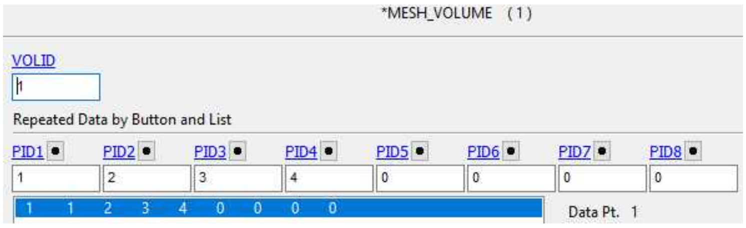
Set Freeslip condition on the upper and lower boundary and Nonslip condition on Wing Profile itself.

- **Keyword Manager → ICFD_Boundary_Prescribed_Freeslip→ PID=3 → Accept**
- **Keyword Manager → ICFD_Boundary_Prescribed_Nonslip→ PID=4 → Accept**

Volume Mesh

In order to create the volume mesh of the fluid domain, the ICFD solver selects four existing parts as follows. The element layer generation of the boundary layer can be set by * **Mesh_BL**. If three layers are set, three-element layers are created on the interface as shown in the figure below, so it is easy to analyze the fluid behavior on the interface.

- **Keyword Manager → Mesh_Volume → select 4 part, insert → Accept**
- **Keyword Manager → Mesh_BL → NELTH=3 → Accept**



*MESH_VOLUME (1)

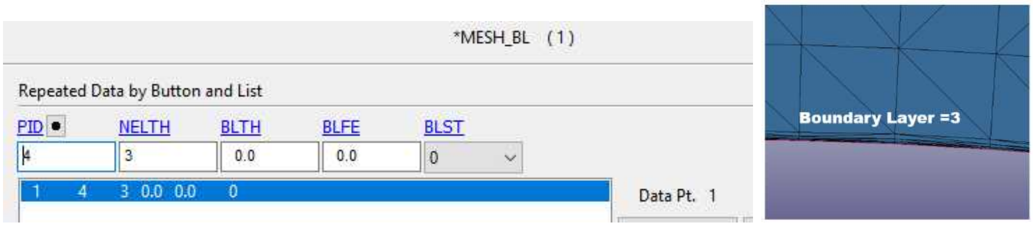
VOLID
1

Repeated Data by Button and List

PID1 ●	PID2 ●	PID3 ●	PID4 ●	PID5 ●	PID6 ●	PID7 ●	PID8 ●
1	2	3	4	0	0	0	0

1 1 2 3 4 0 0 0 0

Data Pt. 1



*MESH_BL (1)

Repeated Data by Button and List

PID ●	NELTH	BLTH	BLFE	BLST
4	3	0.0	0.0	0

1 4 3 0.0 0.0 0

Data Pt. 1

▲ Boundary Layer setup for ICFD model

Analysis Condition

Since the analysis time is 0.005sec, enter **Termination** = 0.005 and set the **d3plot** and **ASCII** data extraction settings at 5e-5 intervals in the output database. In the case of nonlinear analysis such as Bird collision, the input of Bulk Viscosity can improve the analysis efficiency. Input Quadratic coefficient (Q1) and Linear coefficient (Q2) of Viscosity, respectively.

- **Database | Binary_D3PLOT → DT=0.1 → Accept**
- **ICFD_Control_Time | TTM=1 → Accept**
- **ICFD_Control_Output | MSGL=4, LSPPOUT=1 → Accept**
- **ICFD_Database_Drag | PID=4 → Accept**

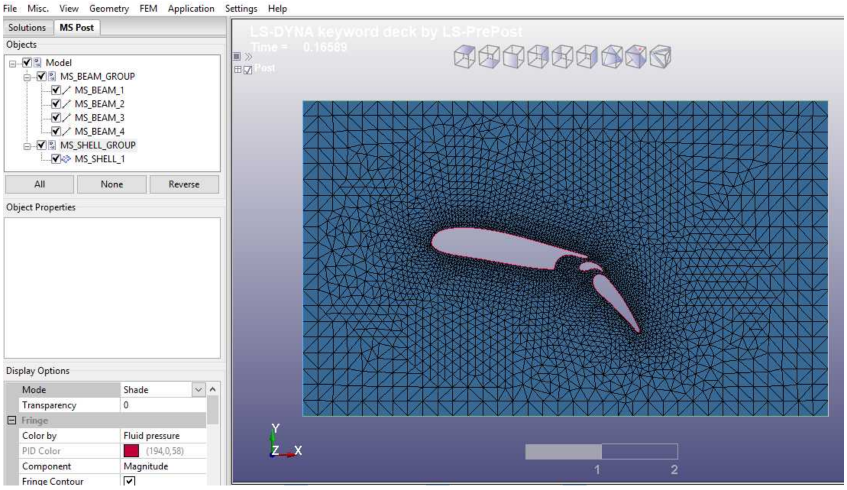
This concludes the analysis modeling of 2D Wing Airfoil for the ICFD solver. Save the completed model as W10_SETUP_2DWing.k and run the analysis. When interpreting ICFD, you must return the interpretation with double precision. If you are a student using the Command window, you can use the following keywords. Readers using the Run LS-DYNA tool run the solver by selecting a double-precision solver with _d.

```
>> LSDYNA –dp i=w10_SETUP_2DWing.k
```

In general, double precision takes 30% longer than single precision. Since this exercise model is a 2D model, it will take about 10 to 11 hours to analyze in a good PC system or workstation.

Result Visualization

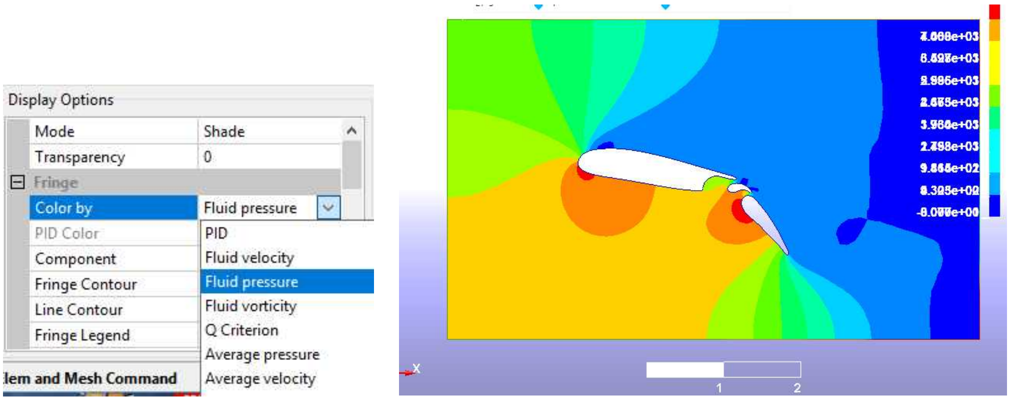
Load **D3plot** into Prepost. You can check the mesh of the fluid region created by the ICFD solver as shown below. In this workshop, we used a relatively large mesh, but it may be a good idea to analysis it with a finer mesh and see what would be the difference in the results. On the left side, you can see the post-processing option for the multi-solver called **MS Post**, and inside you will see the **MS_Beam and MS_Shell** elements created during the modeling process.



Mode: Shade
Transparency: 0
Color by: Fluid pressure
PID Color: (194,0,58)
Component: Magnitude

▲ D3Plot result

Select **Object** = **MS_Shell** from the left option menu and select the drop-down menu of Color by in the Display option at the bottom. Let's choose **Fluid_pressure**. If you look at the **Animate** bar, it will contain more than 1000 frames. The number of frames calculated by the ICFD solver taking into account the characteristics of the fluid and clicking Forward shows that the pressure distribution changes fairly smoothly. The maximum pressure would have been calculated to be about 5 kPa at the leading edge.

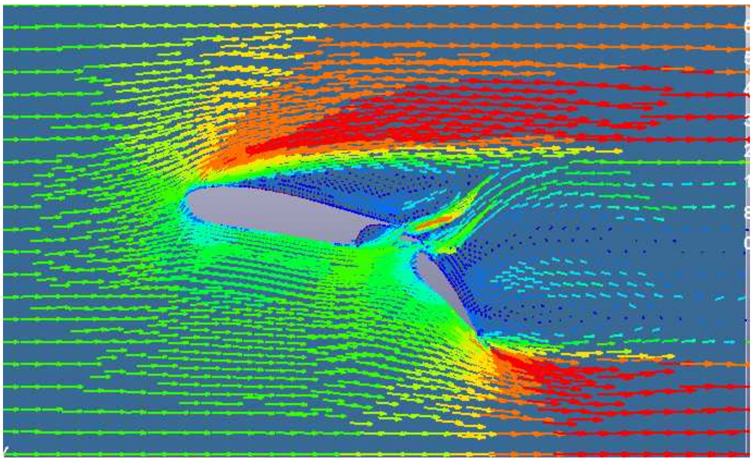


Mode: Shade
Transparency: 0
Color by: Fluid pressure
Component

▲ Fluid pressure distribution

Let's also visualize other variables such as fluid velocity and vorticity. To see the streamline of the fluid, select **Vector** from the Post options on the right and select **Fluid velocity**. As shown in the figure below, it shows the flow of vector which is easy to analyze the flow pattern.

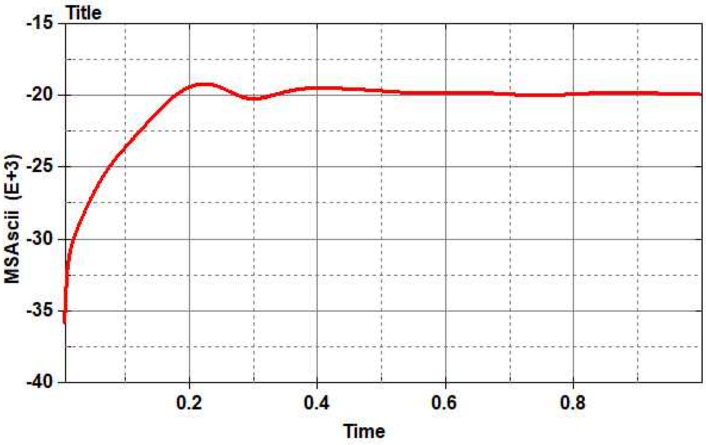
- **Post → Vector | Fluid Velocity: ICFD → Dynamic, SF=0.5, Whole → Apply**



▲ Fluid velocity vector visualization

Let's plot the total drag force occurring on the wing surface using **Ascii** in **MSPost**. **Icfdraga** means the average value of the pressure and velocity distributions, and **icfdragi** is the value extracted directly from the surface element and integrated. Over time, you can see the tendency to converge to a specific value.

- **MS Post → Ascii → icfdraga, Load → icfdraga → Ftot (Fp+Fv) → plot**



▲ Drag force along time

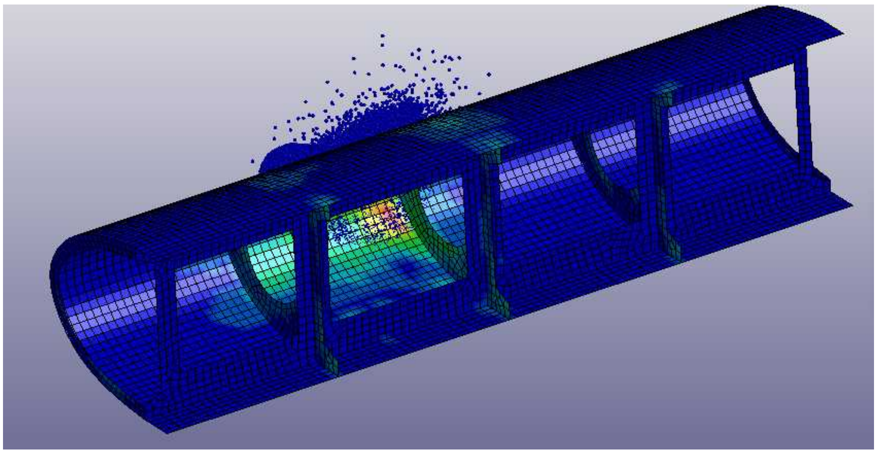
So far, we have practiced the 2D modeling technique using Wing Profile to utilize basic ICFD solver. The 3D modeling procedure is very similar, too. Even readers whose main task is structural analysis should be interested in the analysis of various physical phenomena such as fluid and heat transient analysis because multiphysics analysis is necessary for most industries. In particular, the ICFD field is an indispensable field for structure and fluid interaction analysis (FSI), so it is recommended to learn constant interest.

Part C

Experienced Level

In this part, you will learn and practice more complex analysis using the unique characteristics of LS-DYNA, such as meshless methods or explosion analysis, beyond the scope of collision analysis on general structures.

Chapter 11. SPH Bird Strike Analysis



Workshop Objectives

- ✓ **Learning the concepts of smooth particle hydrodynamics**
- ✓ **Modeling Bird Properties of Soft Materials (* EOS)**
- ✓ **Learning the Bird Strike Modeling Procedure**

Workshop Duration: 80 min

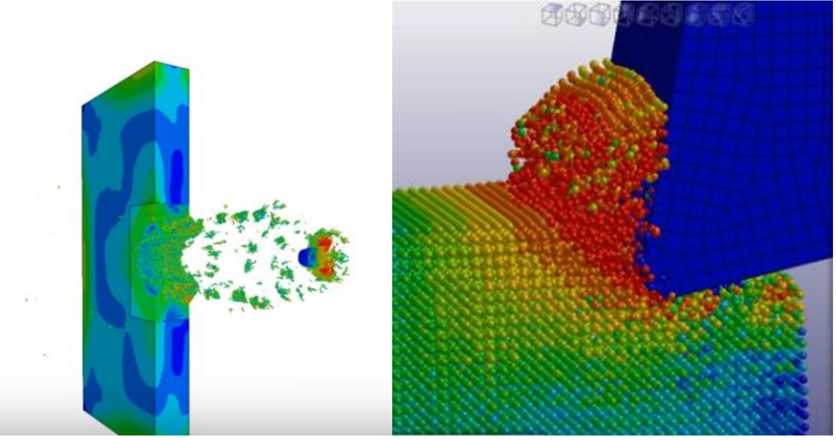
Application: Aerospace, SPH, Bird Strike Analysis

Model Files

- W11_BirdGeom.bdf: Bird mesh model
- W11_Initial_Birdstrike_Mesh.k: Initial state of model
- W11_SETUP_SPH_Birdstrike.k: Final state model (Initial condition)
- W11_SETUP_SPH_Birdstrike_thick.k: Final state (Thickness up)
- W11_SETUP_SPH_Birdstrike_multi.k: Final state (Multi-Bird type)

Introduction of SPH(Smooth Particle Hydrodynamics)

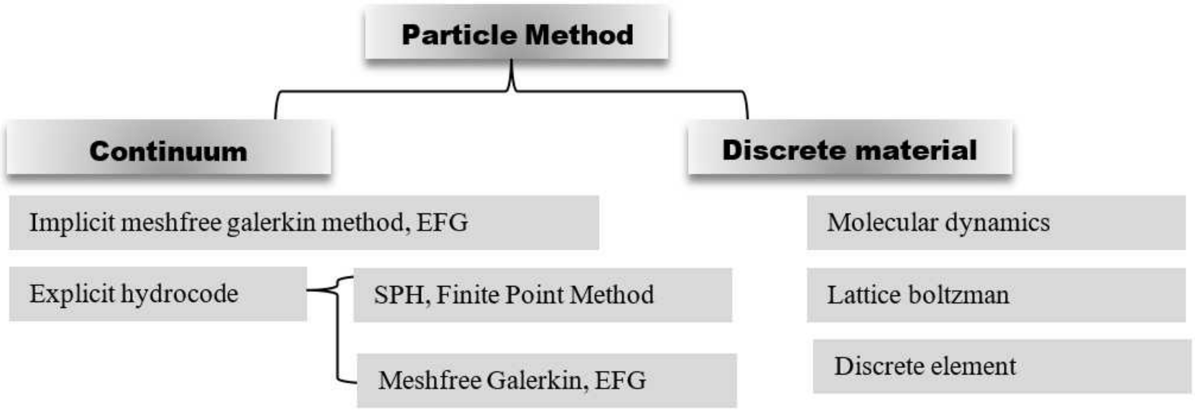
Smooth Particle Hydrodynamics (SPH) is a meshless or mesh-free method for analyzing the dynamic behavior of a continuum. The meshless method enables the modeling of physical phenomena such as bird strikes, sloshing, and compressible fluid analysis to be very close to the real physical world. The SPH technique was proposed by Gingold, Monaghan, and Lucy in 1977, was initially applied to solve the problem of the astrophysical problem.



▲ Various SPH application cases (high-speed collision, processing field)

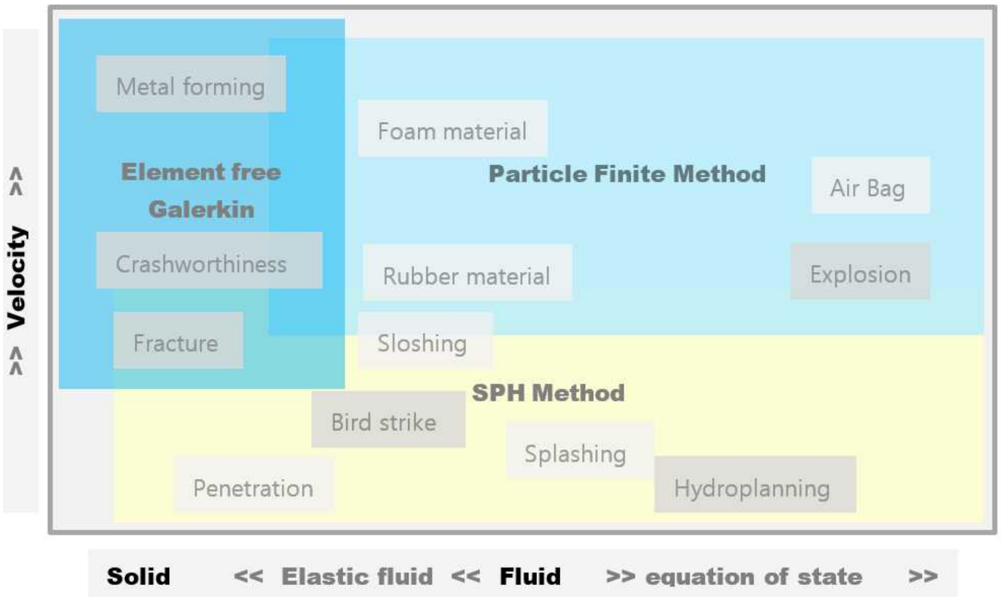
- **Meshfree Method**

There are several analysis techniques for analyzing large deformations or highly ductile materials. One of them is the SPH technique that we are introducing here in this workshop and there is also a finite point method proposed by Onate in 1996. In addition, the EFG (element free Galerkin) analysis technique is one of the widely used techniques. Also, there are a variety of meshfree or meshless techniques. Like the SPH technique, the procedure of modeling an object with particles is also called the Particle Method. The SPH technique is classified as a representative Hydrocode and most commercial FEM programs support Hydrocode.

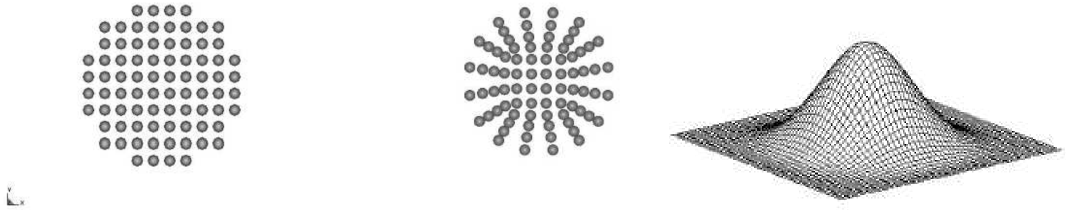


▲ Particle Method Classification

The scope of this mesh-free technique varies considerably from gas to liquid and solid. The chart below shows the concept of the scope of the SPH method among the meshfree methods.



SPH technique is a meshless Lagrangian technique. Individual SPH particles act as interpolation points and the smoothing length of neighboring particles is calculated between individual particles to simulate particle behavior using kernel functions. Basically, a basic mesh model should be created for SPH generation. Due to the characteristics of the SPH concept, the distance, volume, and density between particles must be uniformly configured in the part for stable calculation. Therefore, the numerical error can be reduced by the use of a uniformly distributed model(first case) as shown in the below figure. The shape function of the SPH is in form of a hat shape, which defines the characteristics of the variable for spacing between particles. For example, the force will be higher as it is closer to the particle and smaller as it moves farther away.



▲ SPH Particle Structure Comparison and Shape Function

- **SPH keyword**

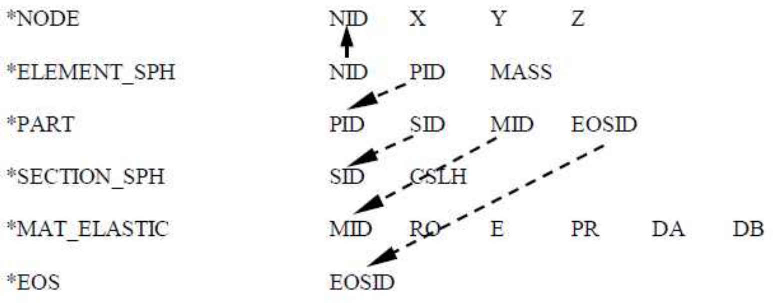
  ***CONTROL_SPH:** Set basic parameters for calculation between SPH particles

  ***SECTION_SPH:** Define the properties of the part containing the SPH particles

  ***ELEMENT_SPH:** Define the mass of SPH particles and the parts they belong to

As shown in the figure below, the SPH particles, parts, SECTION, and materials are set as the basic components. Each particle is recognized as a node and the nodal force, energy, and stress between particles are calculated by the law of conservation of energy, mass, and momentum, and the trajectories and various output values of individual particles can also be extracted.

The application of this SPH technique not only solves the problem of large deformation such as crashworthiness and hypervelocity but also helps in solving fluid-structure interaction with constantly changing boundary conditions and metals processing such as forging and extrusion. Also, highly non-linear problems such as molding process and crack propagation can be easily solved. However, because the computation must be performed on each particle, it takes quite a long time and often requires high computational resources. In addition, if the spacing or physical properties between particles are not modeled correctly, numerical errors may occur.



▲ Relationship between SPH Particles and Part Settings

Various methods are provided below for the SPH application. If you don't need to consider the special conditions, then it will be calculated reasonably with the default values.

- **SPH Formulation Method :**

  0: standard formulation (default)

  1 : renormalized formulation

  2 : symmetric formulation

  3 : symmetric formulation with renormalization

  4 : elliptical formulation

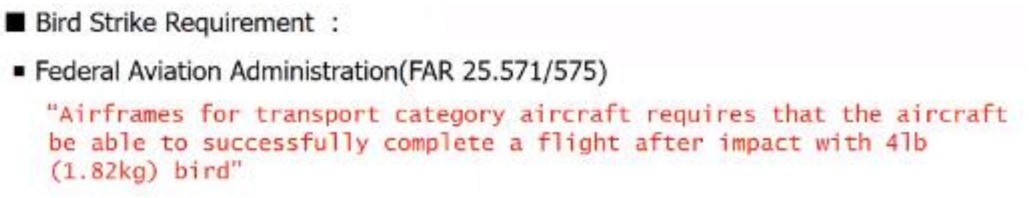
  5 : fluid formulation

  6 : fluid formulation with renormalization

W11 Workshop 11.
Bird Strike Analysis of Aircraft Leading Edge

Introduction

The collision of birds on an aircraft while cruising in the air is called Bird Strike event and it is one of the most dangerous accidents that often occur. Therefore, certification of a bird strike is one of the important processes in aircraft design. In particular, for civil aircraft, there is a specific requirement FAR25.571 for bird impact as specified by the Federal Aviation Administration (FAA). Even if a 4lb bird collides, the flight should able to complete normally and the aircraft should not be damaged.

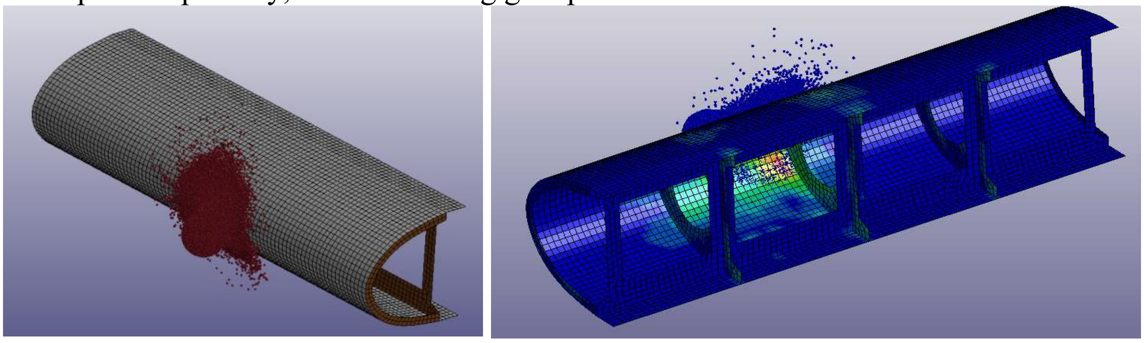


■ Bird Strike Requirement :

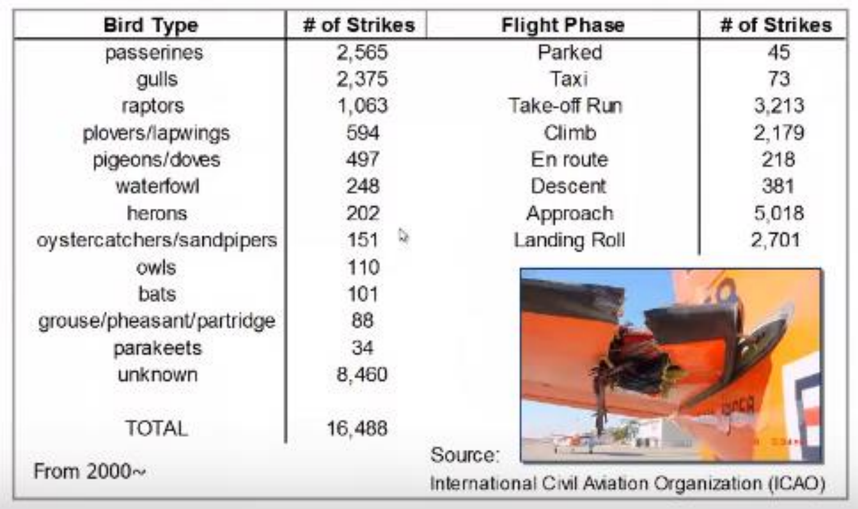
- Federal Aviation Administration(FAR 25.571/575)

  "Airframes for transport category aircraft requires that the aircraft be able to successfully complete a flight after impact with 4lb (1.82kg) bird"

Therefore, in order to certify the structural integrity of a bird strike, it is necessary to verify experimentally and analytically, the airworthiness certification of the Aircraft for Bird impact dynamics. The below table shows the statistics of bird strikes for about 10 years since 2000, published by the ICAO. The data reveals numerous bird strikes were reported, sometimes leading to catastrophic accidents. In particular, the areas where bird strikes occur a lot are on the front leading edge of the wing, the cowling part of the engine, rotating engine fan blades, the cockpit transparency, and the landing gear part.



▲ Analysis result of the bird crash training model



Bird Type	# of Strikes	Flight Phase	# of Strikes
passerines	2,565	Parked	45
gulls	2,375	Taxi	73
raptors	1,063	Take-off Run	3,213
plovers/lapwings	594	Climb	2,179
pigeons/doves	497	En route	218
waterfowl	248	Descent	381
herons	202	Approach	5,018
oystercatchers/sandpipers	151	Landing Roll	2,701
owls	110		
bats	101		
grouse/pheasant/partridge	88		
parakeets	34		
unknown	8,460		
TOTAL	16,488		

From 2000~

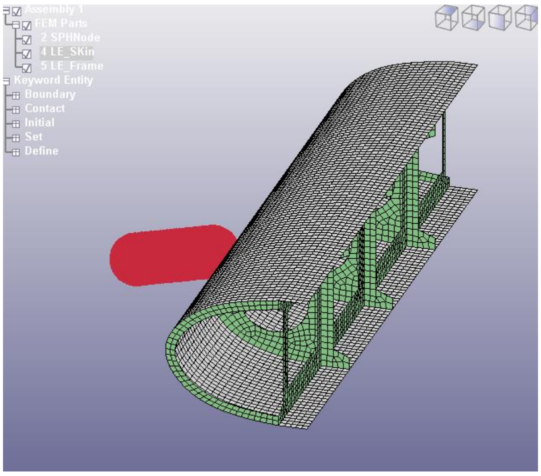
Source:
International Civil Aviation Organization (ICAO)

▲ Current Status of Bird Crash (10 Years Since 2000)

The analysis of large deformations such as the bird strike has many limitations to simulate the actual phenomena with the conventional FEM technique such as the Lagrangian element. With the expansion of dynamic explicit codes such as PAM-CRASH and LS-DYNA in the late 90s, many researchers have investigated on many numerical analysis models for bird strikes, including techniques that allow large deformation analysis such as ALE (Arbitrary Lagrangian element) and SPH. Many studies have been carried out to verify the results of these methods. The modeling of bird strikes has been built at a reasonable level at present. Recently, theoretical and experimental studies have been carried out using this meshless technique for different structures.

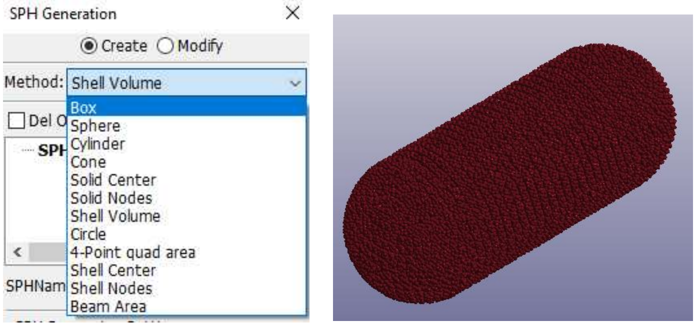
Importing Mesh Model

First, import the model W11_Initial_Birdstrike_mesh.k to analyze the bird strike. As shown in the figure below, you can see the model of the bird created by SPH and the model is made of three components: SPH Bird, the wing leading edge and its inner frame. In order to model bird, a cylindrical shape with rounded ends was applied and SPH particles can be generated through **Element and Mesh→ SPH** generation in LS-Prepost.



▲ Components of Bird Strike Model

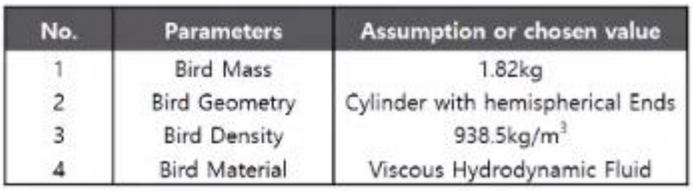
Simple shapes such as cylinders, spheres, and boxes can be created directly in LS-Prepost and in the case of complex geometry, special functions (like Solid Center, Solid Nodes, Shell Volume, etc.) can be used to create SPH with the help of imported geometry. SPH particles with Solid Nodes can be generated using the provided geometry file (* .bdf).



▲ SPH Generation option

The bird SPH model was created using the Solid Nodes option, which generates SPH particles at the node points of the provided geometry model. The mass of the bird part is about 1.82kg according to the FAR regulation and the mass can be calculated based on the density input (Density, 1600kg / m^3). The basic properties are as follows and are assumed to be similar to the average seagull which is about 1-1.8kg. So with this size in mind, it's helpful to model SPH bird.

Bird species-Sea gull (1kg~1.8kg)



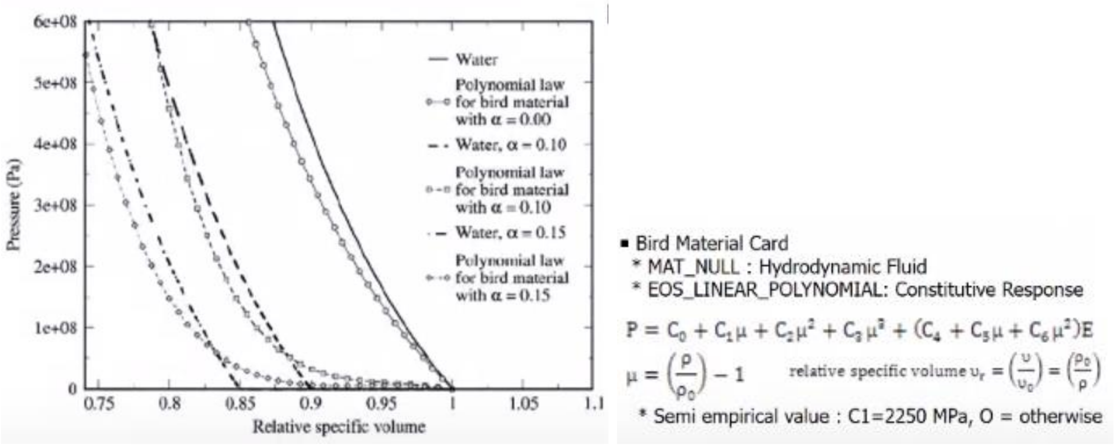
No.	Parameters	Assumption or chosen value
1	Bird Mass	1.82kg
2	Bird Geometry	Cylinder with hemispherical Ends
3	Bird Density	938.5kg/m^3
4	Bird Material	Viscous Hydrodynamic Fluid

* FAA standard requirement mass for bird strike : 4 lb(1.82kg)

▲ Properties of Bird Model

Material Properties

The structure of the leading edge is composed of a thin skin structure composed of a 1.2mm aluminum panel and a 3mm inner-frame. The MAT24 linear plastic property was used and the bird was modeled using NULL properties with EOS (Equation of State). Organic creatures such as birds are composed of internal organs, flesh, feathers, etc., which have bones, beaks and some kidneys having almost no physical properties. The chart below shows the relationship between volume and pressure. It can be seen that the behavior pattern is almost similar to water. EOS is used to express these ductile properties.



- Bird Material Card
  - * MAT_NULL : Hydrodynamic Fluid
  - * EOS_LINEAR_POLYNOMIAL: Constitutive Response

$$P = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu + C_6\mu^2)E$$

$$\mu = \left(\frac{\rho}{\rho_0}\right) - 1 \qquad \text{relative specific volume } \upsilon_r = \left(\frac{\upsilon}{\upsilon_0}\right) = \left(\frac{\rho_0}{\rho}\right)$$

* Semi empirical value : C1=2250 MPa, O = otherwise

▲ Comparison of EOS (Water vs. Bird)[8]

NOTE EOS (Equation of State)

The equation of state (EOS) is a formula that describes a particular state of matter and is used primarily to express the correlation with the pressure, volume, temperature or energy of a liquid or gas. Representative state equations learned in schools include Boyle's Law (1662), which states that the equation defines the relationship between gas pressure and volume and Charles Law (1787), which defines the relationship between temperature and volume and finally the Ideal Gas Law (1834).

In LS-DYNA, EOS is applied to accurately simulate the behavior of a material. Pressure can be expressed as a function of density, energy, or temperature. In the physics at which a very high strain rate occurs, many of these EOS are applied. EOS can be applied to SPH particles and among shell elements, it can be applied to types 13, 14, 15 and *element_solid.

---

[8] Reference: J. S. Wilbeck, "Impact behavior of low strength projectiles," DTIC Document, 1978

Typical EOS representing physical properties close to liquid are * EOS_Linear_Polynomial and * EOS_Gruneisen. * EOS_JWL, which is used to analyze explosives, is one of the widely used keywords. The material that replaces the bird applied in this exercise is defined as *EOS_ Linear_Polynomial. Through experimental data, C1 = 2250MPa is obtained and all other constants are set to 0. Such polynomial EOS can be effectively used when the strain rate is not very high. The vehicle can be used properly up to the low-speed collision range of the vehicle and reliability must be sufficiently verified for speeds higher than that.

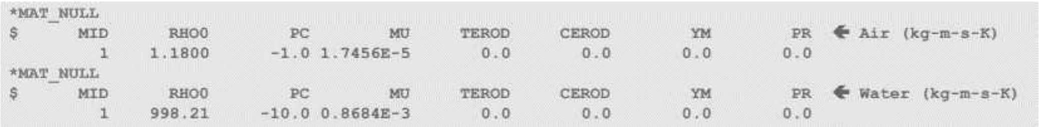
Linear Polynomial equation ▶

$$P = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu + C_6\mu^2)E$$

Gruneisen equation ▶

$$P = \frac{\rho_0 c_0{}^2 \mu \left[1 + (1 - \frac{\gamma_0}{2})\mu - \frac{a}{2}\mu^2\right]}{\left[1 - (s_1 - 1)\mu - S_2 \frac{\mu^2}{\mu + 1} - S_3 \frac{\mu^3}{(\mu + 1)^2}\right]^2} + (\gamma_0 + a\mu)e$$

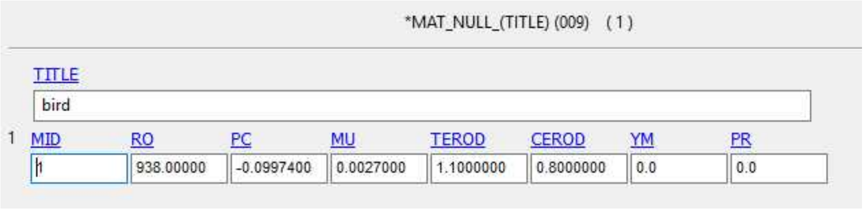
For fluid-like materials (Bird) covered in this exercise, apply * MAT_NULL with * EOS. The MAT_NULL properties of Air and Water are as follows. RHOO stands for density and Mu stands for dynamic viscosity (Pa * s). In other words, it is used together with EOS to calculate the stress acting on the object by calculating the deviatoric stress and pressure.



```
*MAT_NULL
$      MID      RHOO         PC        MU     TEROD     CEROD        YM        PR  ← Air (kg-m-s-K)
         1    1.1800       -1.0 1.7456E-5       0.0       0.0       0.0       0.0
*MAT_NULL
$      MID      RHOO         PC        MU     TEROD     CEROD        YM        PR  ← Water (kg-m-s-K)
         1    998.21      -10.0 0.8684E-3       0.0       0.0       0.0       0.0
```

The linear polynomial parameters of ideal gas consists the following value: rho = 1.025 kg / m3, γ = 1.403, C4 = 0.403, C5 = 0.403. And the Gruneisen constants of water are rho = 1000kg / m3, C0 = 1490m/s, S = 1.79, Γ0 = 1.65. Bird properties are modeled using ***EOS** and ***MAT_NULL** as previously described. Aluminum, the leading edge skin, and frame material input the properties of ***MAT24 Linear Plasticity**. In this case, the fail strain is the criterion to simulate the element erosion that removes the failed element.

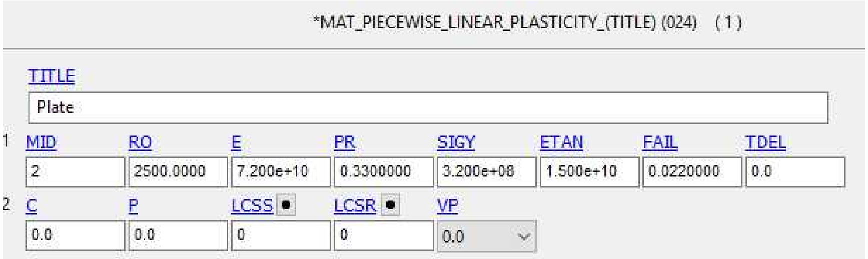
- **Keyword Manager → MAT | 009-NULL → input properties (see image)**
- **Keyword Manager → MAT | 024-Piecewise_Linear_Plasticity → input properties**
- **Keyword Manager → EOS | Linear_Polynomial → C1=2250MPa, remaining is 0**



*MAT_NULL_(TITLE) (009) (1)

TITLE: bird

	MID	RO	PC	MU	TEROD	CEROD	YM	PR
1	1	938.00000	-0.0997400	0.0027000	1.1000000	0.8000000	0.0	0.0

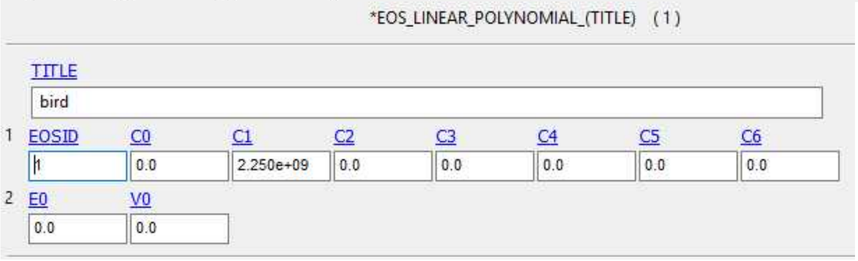


*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE: Plate

	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
1	2	2500.0000	7.200e+10	0.3300000	3.200e+08	1.500e+10	0.0220000	0.0

	C	P	LCSS	LCSR	VP
2	0.0	0.0	0	0	0.0



*EOS_LINEAR_POLYNOMIAL_(TITLE) (1)

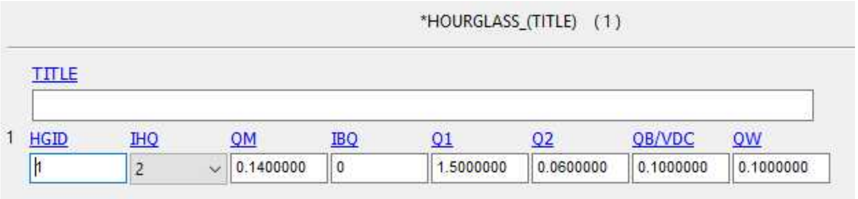
TITLE: bird

	EOSID	C0	C1	C2	C3	C4	C5	C6
1	1	0.0	2.250e+09	0.0	0.0	0.0	0.0	0.0

	E0	V0
2	0.0	0.0

Large deformation occurs in the bird strike analysis, so set hourglass as below and set it in the SPH part. Hourglass is set to the Flanagan-Belytschko viscous form (IHQ = 2) for the SPH part.

- **Keyword Manager → HOURGLASS → IHQ=2, Input (see below)**



*HOURGLASS_(TITLE) (1)

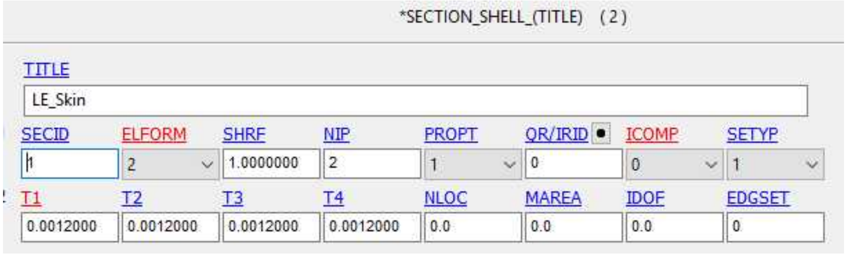
TITLE:

	HGID	IHQ	QM	IBQ	Q1	Q2	QB/VDC	QW
1	1	2	0.1400000	0	1.5000000	0.0600000	0.1000000	0.1000000

Element Properties

After defining the properties, the properties should be defined for the target part. First, **Section_shell** of leading edge consisting of the Shell element is defined as follows. Create a section of 1.2mm for the leading edge skin structure and 3mm for the inner frame. Then define the section of the SPH. **CSLH** is a constant that defines the relative distance between particles. In general, 1.2 is acceptable. **Hmin** and **Hmax** are the variables that define the range of particles taking into account when it moves around. Therefore, **Hmin** 0.2 and **Hmax** 2.0 are input.

- **Keyword Manager → Section_Shell → ELFORM=2, NIP=2, Thickness= 1.2mm**



*SECTION_SHELL_(TITLE) (2)

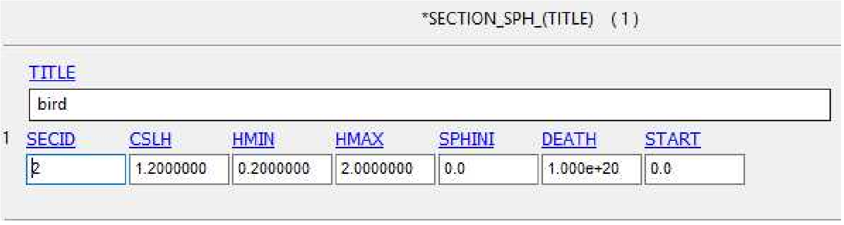
TITLE: LE_Skin

SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	SETYP
1	2	1.0000000	2	1	0	0	1

T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
0.0012000	0.0012000	0.0012000	0.0012000	0.0	0.0	0.0	0

* Similarly, create another 3mm section.

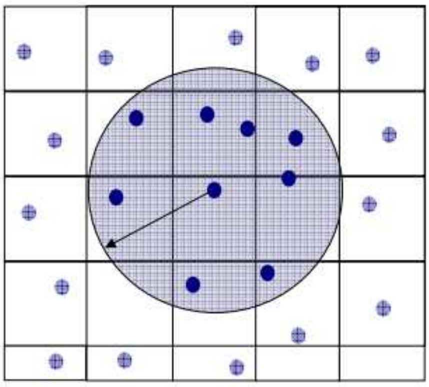
- **Keyword Manager → Section_SPH → CSLH=1.2, see the below image**



*SECTION_SPH_(TITLE) (1)

TITLE: bird

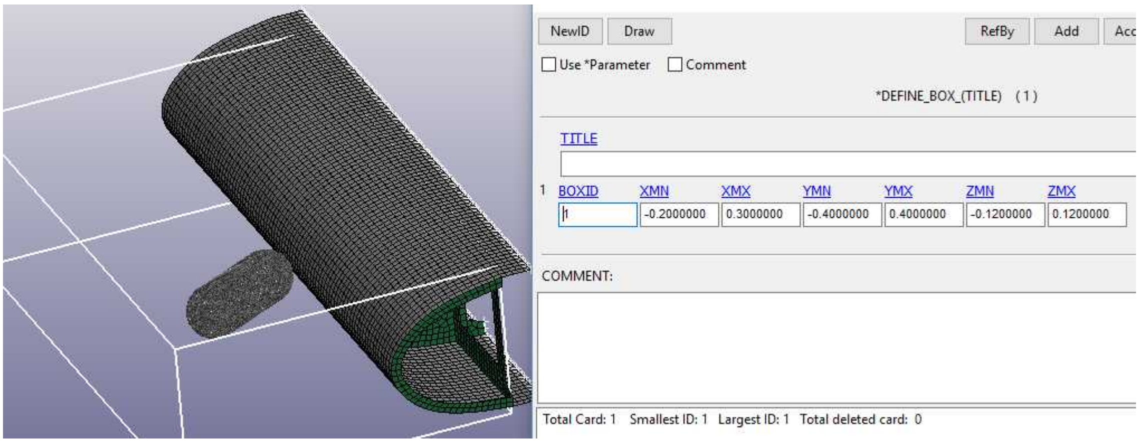
SECID	CSLH	HMIN	HMAX	SPHINI	DEATH	START
2	1.2000000	0.2000000	2.0000000	0.0	1.000e+20	0.0



▲ Concept of calculation range of adjacent particles of SPH particles

When the collision between SPH particle and leading edge occurs, the calculation time of the SPH particle can reduce the analysis time by setting the collision region using **Define_BOX**. Use **Define_BOX** to define the area to consider collision and select BOX from **Control_SPH**.

- **Keyword Manager → Define_Box → Set range values to include collision areas**



*DEFINE_BOX_(TITLE) (1)

TITLE:

BOXID	XMN	XMX	YMN	YMX	ZMN	ZMX
1	-0.2000000	0.3000000	-0.4000000	0.4000000	-0.1200000	0.1200000

COMMENT:

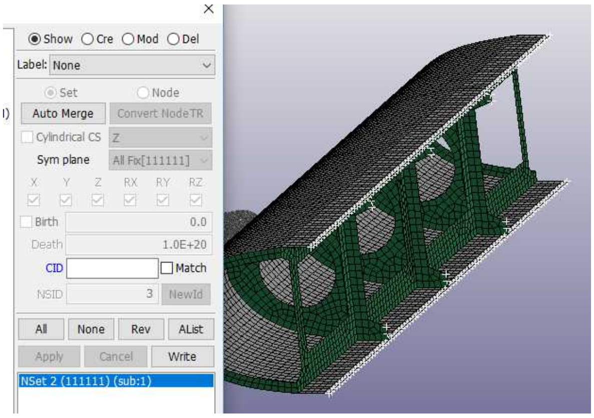
Total Card: 1 Smallest ID: 1 Largest ID: 1 Total deleted card: 0

▲ Define_Box Setting

Boundary Condition

As shown in the figure below, fix all the parts that are connected to the main wing of the leading edge structure. The speed of the SPH part is set to the initial speed with **Initial_velocity**.

- **Model and Part→ Create Entity→ SPC | Cre→ Sel. Node | Area→ Select the node in the back of the leading edge structure as shown below. Check all 6 D.O.F→ Apply**



Label: None

Cylindrical CS: Z

Sym plane: All Fix[111111]

☑ X ☑ Y ☑ Z ☑ RX ☑ RY ☑ RZ

☐ Birth 0.0

Death 1.0E+20

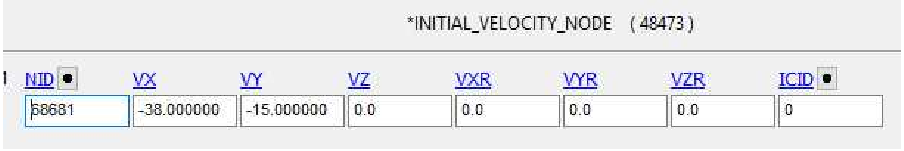
CID

NSID 3

NSet 2 (111111) (sub:1)

▲ Boundary condition Setting

- **Model and Part→ Create Entity→ Initial | Cre→ Sel. Node | Area→ Select all SPH parts→ Enter Vx = -38, Vy = -15**



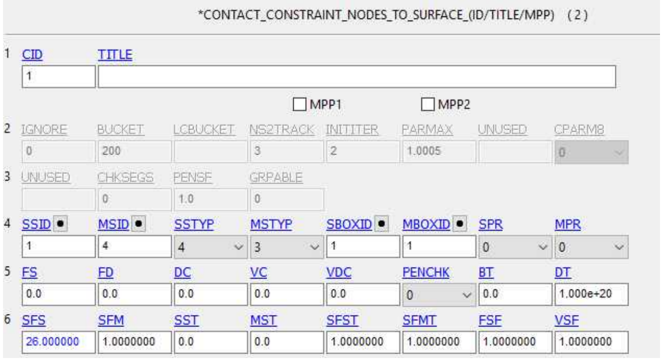
*INITIAL_VELOCITY_NODE (48473)

NID	VX	VY	VZ	VXR	VYR	VZR	ICID
68681	-38.000000	-15.000000	0.0	0.0	0.0	0.0	0

Contact Condition

For contact between SPH particle and shell element, the contact conditions **Surface_to_Nodes** series must be selected. You can use conditions like **Automatic Nodes_to_Surface, Constraint_Nodes_to_Surface,** and **Eroding_Nodes_to_Surface**. According to the research data of other literature available, the results correlated or which can be most similar to the actual test can be obtained under the contact conditions of **Constraint_Nodes_to_Surface**. However, because there are so many variables for each condition, it is important to apply various conditions and to recognize the difference accurately to get reliable analysis results. Define the SPH particle as a node set and set it as a Slave and the Master Surface sets the structure of the collision as Part or Segment. Note that in the case of shell elements, the Normal direction must face the surface where the collision occurs. In the case of bird impact analysis, the coefficient of friction is defined as zero.

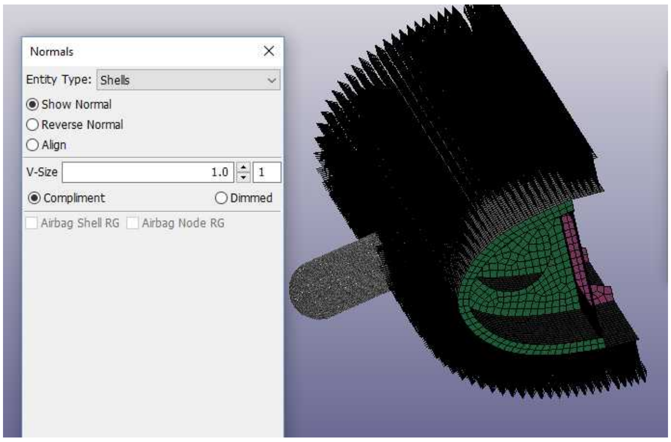
- **Model and Part → Create Entity → Set Data | *Set Node → Cre | SPH select all particles → Apply**
- **Keyword Manager → Contact | Eroding_nodes_to_surface → Master surface defined as Part, Slave Surface defined as Node set**



*CONTACT_CONSTRAINT_NODES_TO_SURFACE_(ID/TITLE/MPP) (2)

1	CID	TITLE						
	1							
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	1	4	4	3	1	1	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.0	0.0	0.0	0.0	0.0	0	0.0	1.000e+20
6	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
	26.000000	1.0000000	0.0	0.0	1.0000000	1.0000000	1.0000000	1.0000000

The normal direction of the shell element should be checked and set in **Element tools → Normals**. Check the Normal direction for any mismatch. If the normal vector of the leading edge's skin does not face the outside as shown below, select **Reverse Normal** to match it as shown below.



Normals

Entity Type: Shells

● Show Normal

○ Reverse Normal

○ Align

V-Size 1.0 1

● Compliment ○ Dimmed

☐ Airbag Shell RG ☐ Airbag Node RG

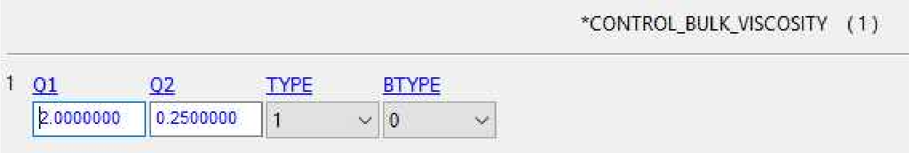
▲ Checking Shell Normal

Analysis Condition

Since the analysis progress time is 0.005sec, enter termination = 0.005 and set the d3plot and ASCII data extraction settings at 5e-5 intervals in the output database. In the case of nonlinear analysis such as bird collision, the input of Bulk Viscosity can improve the analysis efficiency. Input Quadratic coefficient (Q1) and Linear coefficient (Q2) of Viscosity, respectively.

- **Control | Termination → ENDTIM = 0.005, DTMIN=0.25**
- **Database→ ASCII_option → Default DT=5e-5, selet GLSTAT, MATSUM**

- **Database→ D3plot → Default DT=5e-5**
- **Control→ Bulk_Viscosity → input value in Q1 and Q2**



*CONTROL_BULK_VISCOSITY (1)

	Q1	Q2	TYPE	BTYPE
1	2.0000000	0.2500000	1	0

- **Control→ Energy → 2, 2, 2, 2**
- **Control→ SPH → BOXID, IDM=3(3D), Memory= 150**
- **Control → Timestep → TSSFAC=0.5, ERODE=1, DT2MS(Default=0) if necessary**

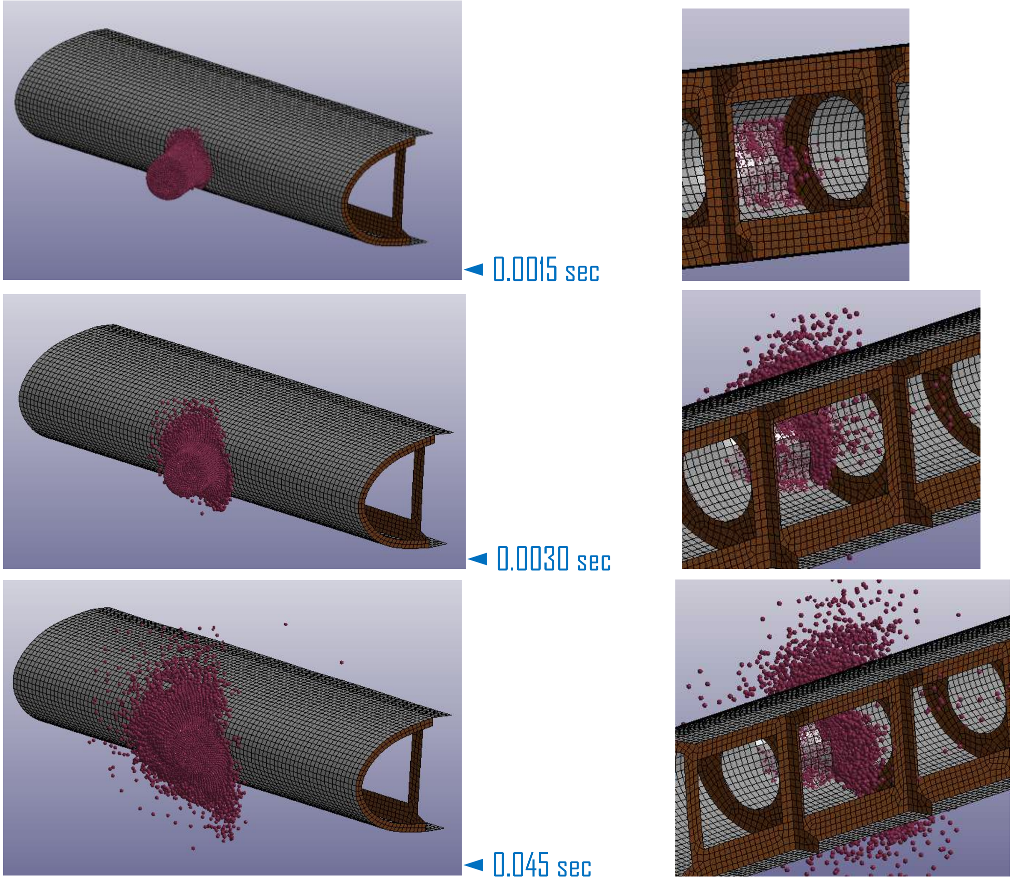
  * Mass scale(DT2MS) is 1e-6 if the analysis takes a long time

Lowering the **TSSFAC** (default = 0.9) to 0.5 is to mitigate the instability that occurs at the contact. Some analysis cases may take longer. Also, select **ERODE** = 1 to enable the element to be removed when the minimum time-step (**TSMIN**) is reached. If the **DTMIN** of the termination is set to 0.25, the initial time-step can be reduced to ¼, thereby eliminating the instability due to the step-size.

This completes the creation of a bird strike model. Using SPH seems to be quite complex and will take a long time to solve, but if you understand the concept, modeling procedure wouldn't be so difficult. In this analysis, the number of SPH particles is about 48,000 and the analysis time taken to solve will be about 1 ~ 2 hours in general PC (i7, 16GB ram). Readers who are burdened with computational time can considerably shorten their analysis time by cutting the number of SPH particles in half.

Analysis Result

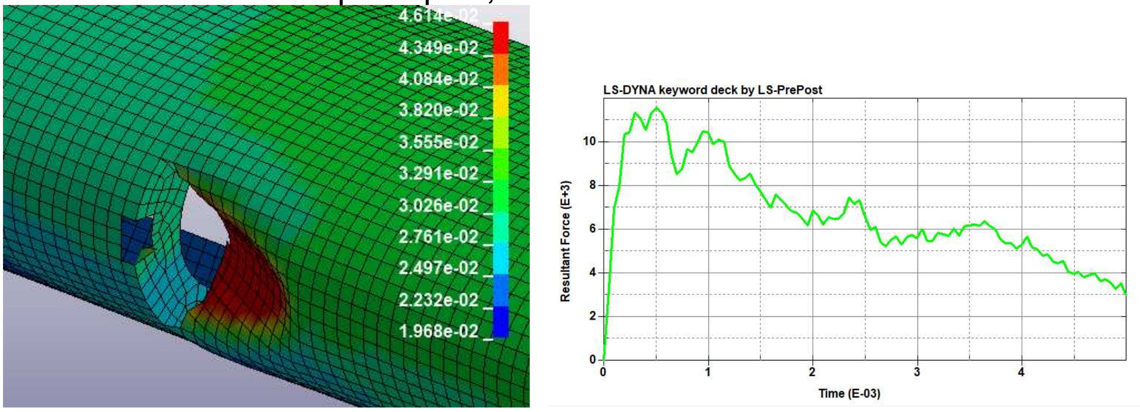
Complete the analysis and load the D3plot into Prepost. Let's check the Bird crash over time. As the impact area of 1.2mm leading edge made of aluminum is damaged by bird collision, element erosion occurs at the region of impact and it can be seen that SPH particles penetrate. In the analysis process, it can be seen that the time step size suddenly decreases as the element erosion occurs. As the contact becomes unstable, it decreases by the previously set DTMIN value and still the analysis after the penetration of the SPH particle continues to solve.



▲ Bird Crash Progression

The impact region was damaged on one side of the area supported by the skin and leading edge Rib due to bird strike. The damage started from the point where the bird SPH part impacted and the maximum impact load was reached. The impact load can be reviewed through the **Rcforce** of the Ascii output as shown.

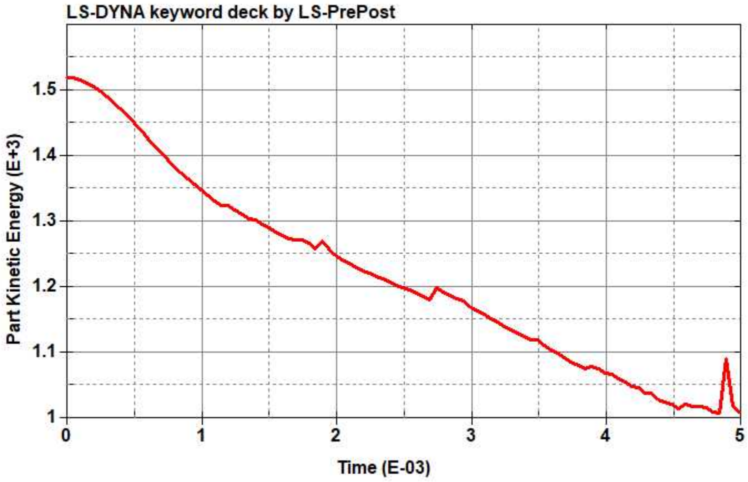
- **Post → Ascii → Rcforce | Load | SI-1, MI-1 → Plot**



▲ Damage site and impact load graph of the impact area

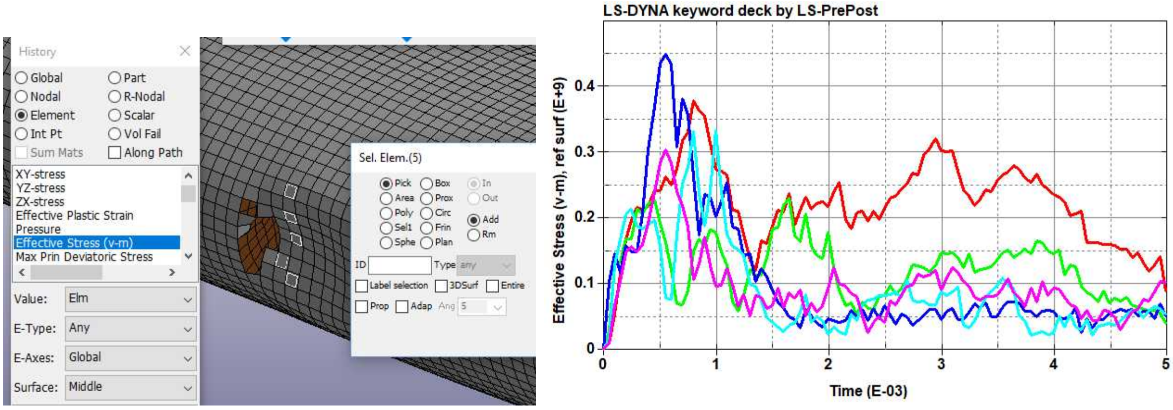
Let's check the kinetic energy of SPH bird parts. It can be seen that gradually decrease from the start of the impact. Also, check the distribution of von mises stress from adjacent elements of the impact area. It is possible to extract the history of the stress at a specific node and element.

- **Post → Hist → Part | Kinetic Energy → select SPH → Plot**



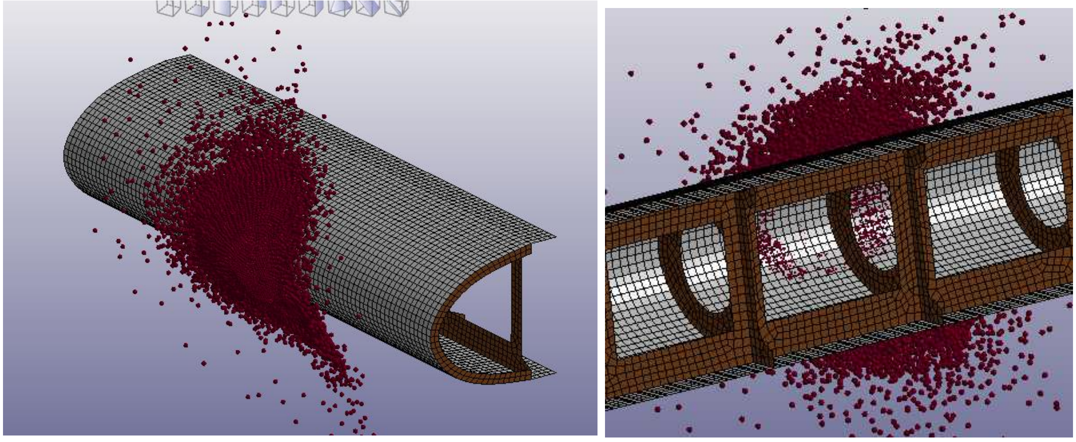
▲ Kinetic energy distribution of SPH part

- **Post → Hist → Element | Effective stress → select some element close to impact site → Plot**



▲ von Mises stress distribution at the impact site

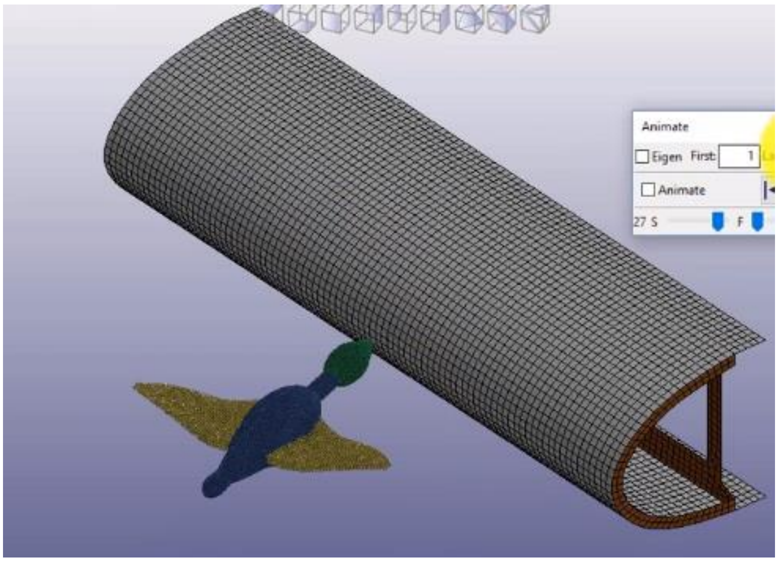
As described above, the impact analysis was conducted under certain conditions. A significant area of the leading edge was torn, affecting flight stability, which does not meet the FAR standard. Therefore, the thickness of the aluminum skin is 1.2mm → 2.4mm. Let's solve the calculation again after doubling its thickness. As shown in the figure below, it can be confirmed that no damage occurred. From the analysis results, the impact load of the impact site, the kinetic energy of SPH particles and the stress of the impact site are extracted and compared with the previous analysis results.



▲ Bird strike of the leading edge with increased thickness

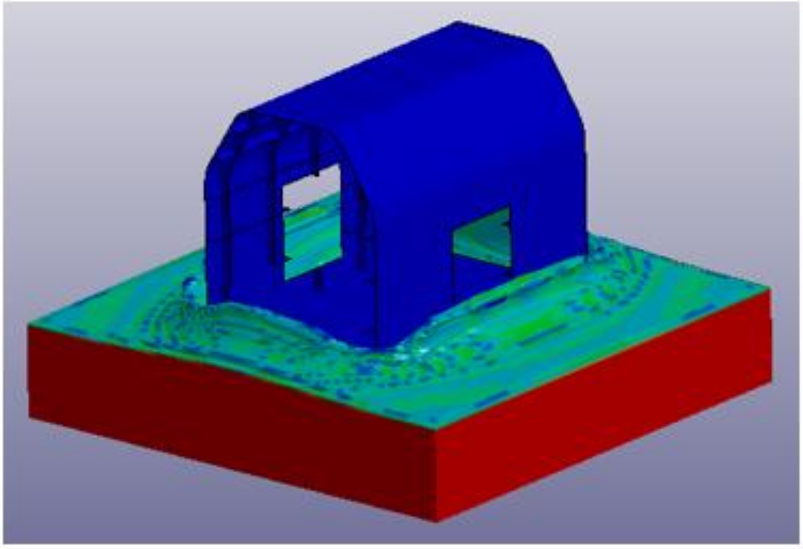
The modeling techniques introduced in this workshop are currently applied to many studies, so they are expected to be useful in various fields. In fact, many experimental results of bird collisions are provided in the research data. The reader should find the optimal modeling value by calibrating various parameters to compare with the experimental data. This parametric study may be one of the main research data because various parameters influence the analysis condition and the situation.

In addition, a model with different physical properties such as head, chest, and wings attached to the SPH main body part can be created to get an ideal shape of a real bird. Perform an impact analysis using this model and compare how it differs from the traditional hemispherical bird model.



▲ Bird strike analysis using realistic bird shape SPH part

Chapter 12. Water Landing Analysis with ALE Method



▌Workshop objectives

✓ **Learning analysis concept of ALE (Arbitrary Lagrangian Eulerian)**

✓ **Learning Water Domain property modeling**

✓ **Learning the application concept of Lagrangian_in_Solid**

✓ **Practice water landing modeling procedure**

▌Workshop duration: 60 min

▌Application: Aerospace, ALE, Water Landing

▌Model file

- W12_Fuselage.bdf: Fuselage mesh model (Nastran)
- W12_Initial_Mesh.k: Initial state of mesh model
- W12_SETUP_Water_ALE_Rigid.k: Final state of model (rigid)
- W12_SETUP_Water_ALE_Elast.k: Final state of model (Elastic)

Introduction of ALE Water Landing

The water landing load imparted on aircraft operating over a sea is one of the important considerations in the structural design. Passenger safety must be ensured during an emergency landing and must be suspended for enough time for the passengers to escape. Until the mid-'90s, there was a regulation to satisfy certain conditions through the mock-up test. Recently, the airworthiness certification is progressing along with the numerical analysis with the development of computer analysis technology. In terms of the aviation industry, numerical analysis can greatly reduce the risk and performance of expensive tests. In this exercise, we shall practice the process of analyzing the water landing load on a simplified helicopter body. Using the Arbitrary Lagrangian-Eulerian (ALE) technique which is the most widely used among the various analysis methods, we will learn how to analyze the water landing load by coupling the Lagrangian structure in the Eulerian water region. This situation of sea crash is called 'Ditching' and the term ditching simulation is used in the aviation field. The photo below shows the Airbus A320-214 aircraft landing in the Hudson River in January 2009, also known as the "Hudson River Miracle," where all 155 passengers were safely rescued. Along with the skilled emergency action of pilot Sullenberger, also known by the film ‘Sully’, the stability of the aircraft's structure also should be contributed.

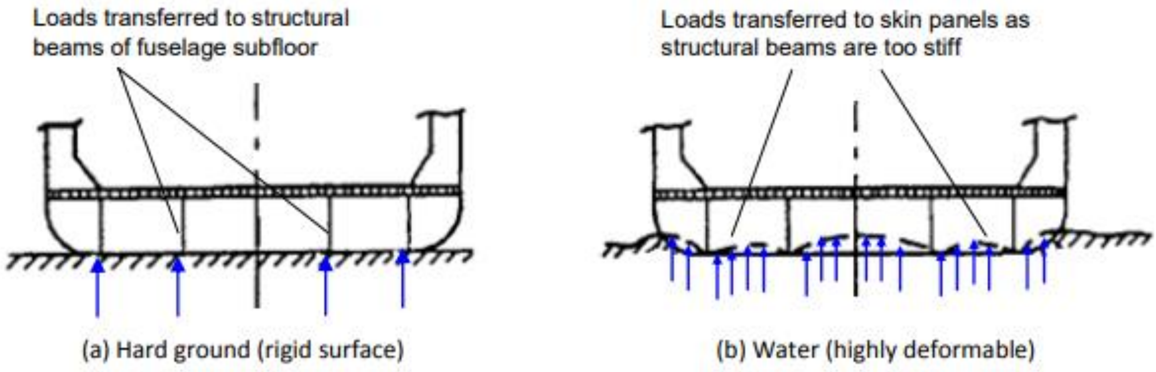
▲ Hudson River Emergency Landing

▲ Water landing load test of the helicopter body[9]

---

[9] Reference: Pentecote N., Vigliotti A., "simulation of the impact of water of a sub-floor component and a fullscale WG30 helicopter", AHS International – The Vertical Society, Forum 58

The above test pictures highlight a water impact test scene conducted by the Aerospace Research Institute of Italy during the development of the Westland G30 helicopter in the UK. WG30 helicopter fuselage is mainly made of aluminum and titanium with a composite sandwich structure applied on to floor structure. It is a test to analyze the degree of structural damage by impacting a full-scale structure weighing about 4300kg at a speed of about 8m / s. Although some aircraft fuselage or helicopter fuselage are performing vertical drop tests in water or above ground in the USA or Europe, there are not many published technical data on the actual water impact data. This workshop will proceed with the Water Impact Analysis using the ALE technique by applying the test conditions similar to the simplified model. Such ditching provisions can be found in FAR 562. Maximum acceleration (14g), speed (31.2ft / s) and displacement (30.3inch) due to impact shock are regulated. Various types of shock absorbing designs are applied to reduce the impact load due to water impact, for example, by inserting a foam absorber into the floor structure, using an absorber such as a shear tie to absorb impact energy, or by using a truss structure. Various types of design elements have been applied to mitigate impact damage. The impact on the ground and the impact on the water have quite different characteristics of behavior. Therefore, ALE (Arbitrary Lagrangian-Eulerian) is a representative technique for large deformation cases rather than the convention FE Mesh model. Basically, in order to understand the ALE technique, the Lagrangian algorithm and Eulerian algorithm should be understood. If you want to understand in-depth, you can see these two concepts in most of the FEM textbooks. In this chapter, only basic concepts will be explained to help you understand the workshop problem.


Loads transferred to structural beams of fuselage subfloor
Loads transferred to skin panels as structural beams are too stiff

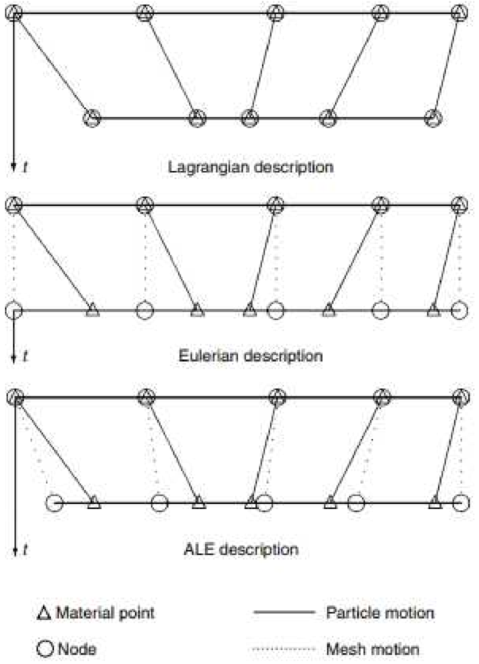
(a) Hard ground (rigid surface)

(b) Water (highly deformable)

▲ Difference between shock load transfer between ground shock and water surface shock

Lagrangian Algorithm

The elements and meshes we often use in structural analysis apply the Lagrangian concept. As individual nodes move together with the movement of the target structure, nodes move together along the location where the deformation of material occurs with time as shown in the figure below. The disadvantage of Lagrangian mesh is that it is quite limited because numerical errors are likely to occur when very large deformation occurs.



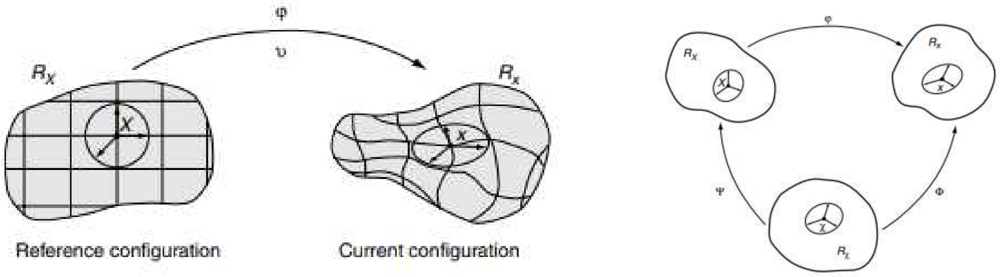
▲ Concept of Lagrangian, Eulerian, and ALE Mesh

Eulerian Algorithm

On the other hand, the Eulerian analysis method is widely used for fluid dynamics. As shown in the above figure, when the material moves over time, the reference point moves with it, so that the large deformation can be relatively easily simulated. The disadvantage is that the definition of the interface of the material is difficult to accurately represent.

ALE(Arbitrary Lagrangian-Eulerian) Algorithm

The ALE algorithm combines the advantages of these two techniques. As shown in the above figure, a node moves like a lagrangian to a specific area and can move freely like a Eulerian mesh within a certain range. Therefore, the large deformation can be simulated relatively easily, and the definition of the interface can be expressed more accurately than the Eulerian mesh. Because of these advantages, they are widely applied because they are very useful in fluid-structure interaction problems.



▲ Concepts of Lagrangian Mesh Movement and ALE Mesh Movement

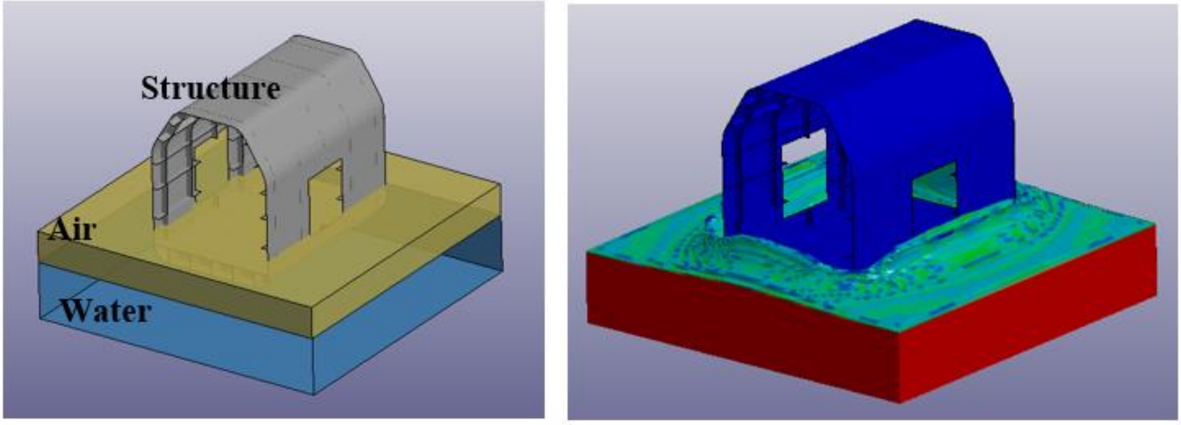
W12

Workshop 12.

Water Landing Analysis of Helicopter Fuselage

Introduction

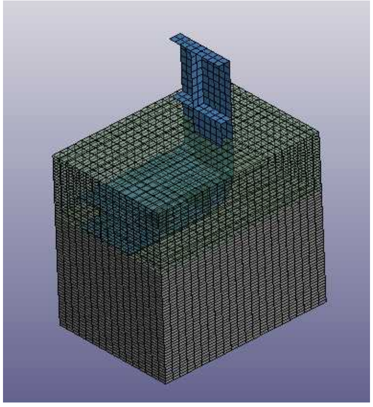
The basic three parts which are needed for the water landing analysis by applying the ALE technique. Representation is made as to the Water domain (Eulerian), Air domain (Eulerian), and Structure (Lagrangian). As shown in the figure below, the helicopter body (shell element), a water domain (solid element), an air domain (solid element).



▲ ALE Water landing model

In this workshop, the simplified helicopter fuselage was modeled as a shell element and analyzed for a vertical drop of 8 m/s. First, we modeled the helicopter fuselage as a rigid body and analyzed the surface pressure of the structural members impacted by the water surface. Next, the modeling was performed with the aluminum elastic body where deformation occurred. The ***Constrained Lagrange_in_Solid** keyword is applied in order to evoke the FSI (Fluid-structure interaction) which helps simulate the interaction between Eulerian water and Lagrangian structure. Because the number of elements is quite large, the whole body analysis requires a considerable time of more than 10 hours in the case of the elastic body analysis. Thus, the partial sub-model was utilized for this workshop to save computational time. The unit system is applied to the meter unit system in consideration of the size of the structure.

- **Unit: m, sec, kg, N, Pa**



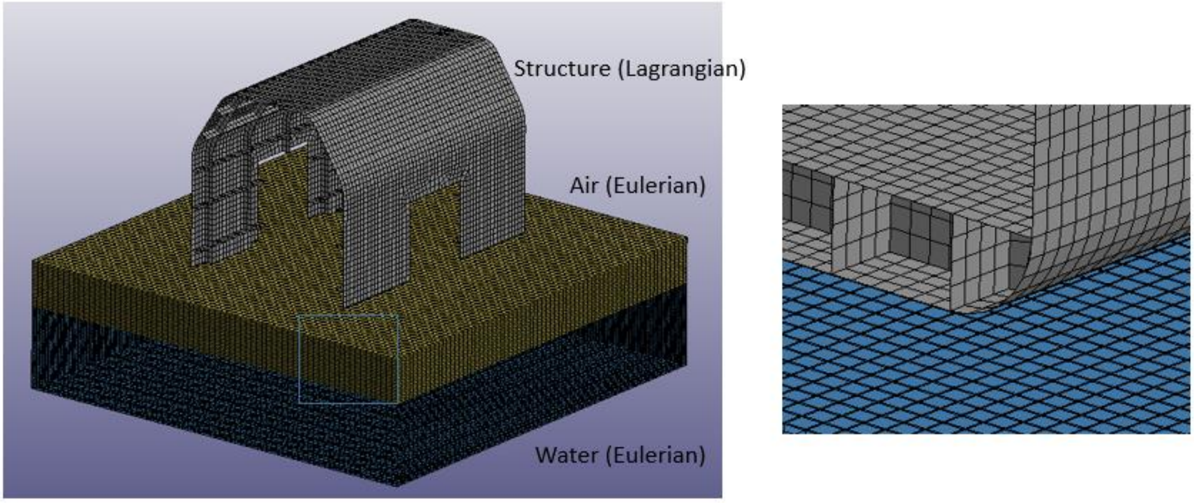
▲ Submodel of ALE water impact model(Elastic body)

Through this hands-on workshop, readers will be able to learn basic single material ALE analysis techniques and understand the overall analysis procedures and application concepts. In addition, the process of extracting the impact pressure acting on the surface of the structure through the analysis process and the post-processing procedure to review the result of the ALE analysis method will be practiced. The ALE analysis technique can simulate the nonlinear physics of large deformations, so it is widely used in various industrial fields.

Mesh Model

Import the model ***'W12_ALEmesh.k'*** which is created for the water landing analysis. The mesh model was created with CAD program and then imported it as a Nastran format (* .bdf). If you import ***W12_Initial_mesh.k***, you will see a three-part model which is as shown below. It is important that the mesh size of the region where the contact between the Lagrange structure and the Euler domain starts should be similar in order to avoid numerical error or severe divergence issue.

There are two ways to apply the ALE method. There is a method of applying the properties of air or water to **Multi-Material ALE (MMA)** and then there is a **single ALE method** of inputting the same properties and setting the air area to Void. In this exercise, the latter method (**Single ALE**) with the air zone set to void was applied. Note that the nodes at the interface between the air and water domains must be shared.

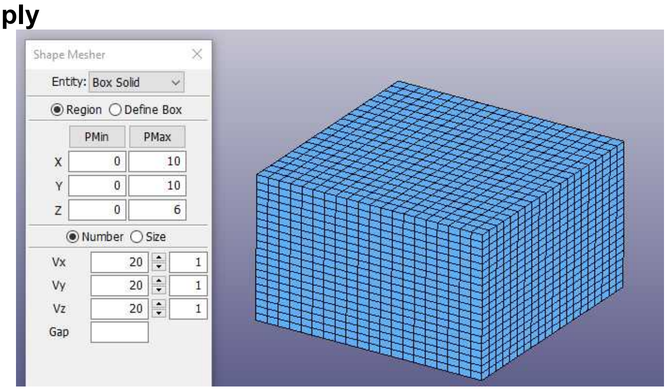


▲ Water landing model having 3 parts

In general, the area of the ALE element should be modeled 3 ~ 5 times larger than that of structural material to reduce the edge effect. However, in this exercise, the ALE domain size is 2 times of structural domain to reduce the analysis time. It is recommended to extend the ALE domain to be used for research work. Creating an ALE region is simple. You can use your own preprocessor, or you can easily create one using LS-Prepost. In LS-Prepost, use the Shaper Mesher's Box to model the domain size including water and air and then use **Move or Copy** to separate the air area and set it as a new part ID. A model that includes only Fuselage (***W12_Fuselage.bdf***) is provided, so please try to do the modeling practice by increasing the ALE domain size after completing this exercise to check the effect of ALE domain size.

- **Shape Mesher → Box Solid → input size and number of the element (see image)**

- **Move or Copy → Move | Target ID=2 → Sel. Elem. | Area | by Element → Select Air by dragging it → Apply**



Shape Mesher

Entity: Box Solid

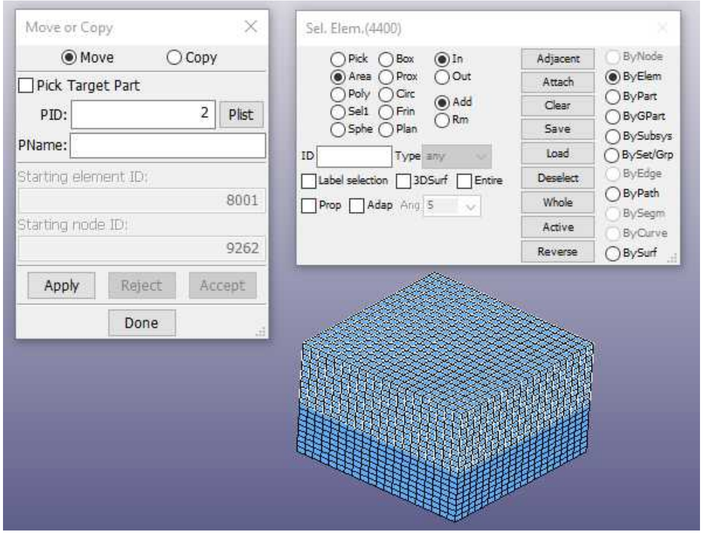
● Region ○ Define Box

	PMin	PMax
X	0	10
Y	0	10
Z	0	6

● Number ○ Size

Vx	20	1
Vy	20	1
Vz	20	1
Gap		

▲ ALE domain creation (Shape Mesher)



Move or Copy

● Move ○ Copy

☐ Pick Target Part

PID: 2

PName:

Starting element ID: 8001

Starting node ID: 9262

Sel. Elem.(4400)

○ Pick ○ Box ● In

● Area ○ Prox ○ Out

○ Poly ○ Circ ● Add

○ Sel1 ○ Frin ○ Rm

○ Sphe ○ Plan

ID Type any

☐ Label selection ☐ 3DSurf ☐ Entire

☐ Prop ☐ Adap Ang 5

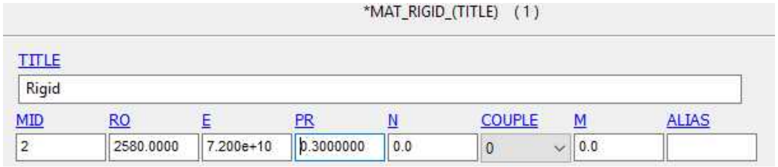
○ ByNode ● ByElem ○ ByPart ○ ByGPart ○ BySubsys ○ BySet/Grp ○ ByEdge ○ ByPath ○ BySegm ○ ByCurve ○ BySurf

▲ Separating water and air domain

Material Property

The material of the helicopter body is made of aluminum and in this workshop, two different properties will be applied and solved to check the difference. We will apply the rigid body properties first and later perform an analysis by applying the elastic body properties where deformation occurs. There are several ways to define the physical properties of the water domain. In this exercise, we modeled using * **MAT_NULL** and * **EOS_Gruneisen**. The water property definition is the most important factor in the analysis results. Please refer to the description of EOS in Chapter 11, Birdstrike.

- **Keyword Manager → MAT | 020-Rigid → Input properties**
- **Keyword Manager → MAT | 009-NULL → Input properties**
- **Keyword Manager → EOS | Gruneisen → Input properties**

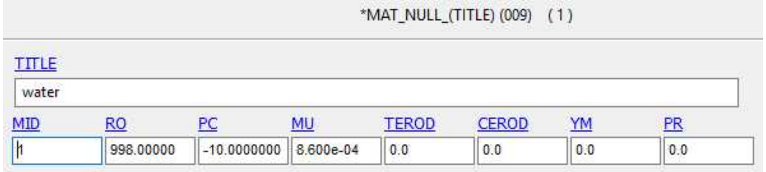


*MAT_RIGID_(TITLE) (1)

TITLE: Rigid

MID	RO	E	PR	N	COUPLE	M	ALIAS
2	2580.0000	7.200e+10	0.3000000	0.0	0	0.0	

▲ MAT020_Rigid Properties

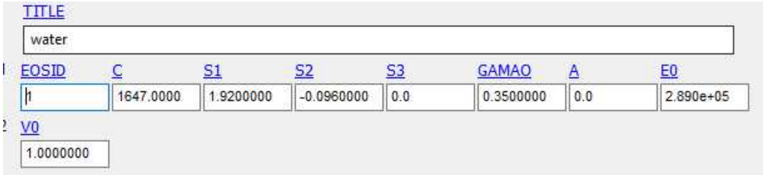


*MAT_NULL_(TITLE) (009) (1)

TITLE: water

MID	RO	PC	MU	TEROD	CEROD	YM	PR
1	998.00000	-10.0000000	8.600e-04	0.0	0.0	0.0	0.0

▲ MAT009_Null Properties



TITLE: water

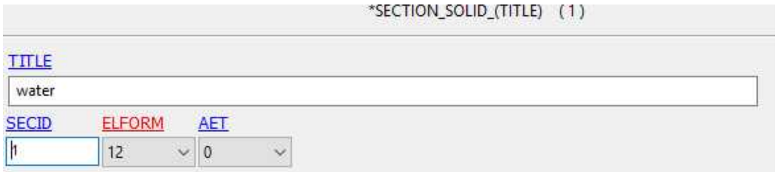
EOSID	C	S1	S2	S3	GAMAO	A	E0
1	1647.0000	1.9200000	-0.0960000	0.0	0.3500000	0.0	2.890e+05
V0							
1.0000000							

▲ EOS_Gruneisen Properties

Element Property

Set the ALE area of Water and Air to Solid element. **ELFORM** can model the same material in various ways according to various analysis techniques. In this exercise, ELFORM is modeled using ELFROM = 12. The fuselage is set using ELFORM = 2 in the shell section.

- **Keyword Manager → Section_Solid → ELFORM=12**
- **Keyword Manager → Section_Shell → ELFORM=2, Thick=0.005**

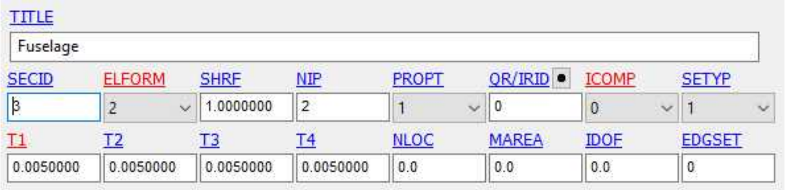


*SECTION_SOLID_(TITLE) (1)

TITLE: water

SECID	ELFORM	AET
1	12	0

▲ Solid section properties for Water and Air Domain



TITLE: Fuselage

SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
3	2	1.0000000	2	1	0	0	1
T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
0.0050000	0.0050000	0.0050000	0.0050000	0.0	0.0	0.0	0

▲ Shell section properties for Fuselage

NOTE Element configuration of ALE domain

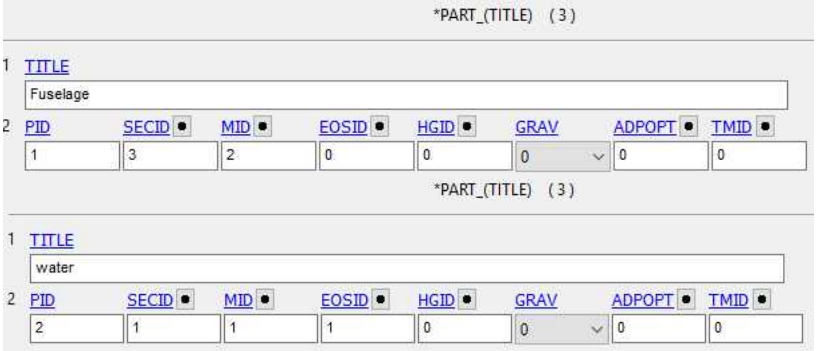
There are various methods of selecting the element formulation of the fluid domain. In the ALE analysis, the water and air areas can be selected as the following ELFORM using 3D solid elements.

- **ELFORM 1 = Constant stress solid (applies to typical Lagrangian models)**
- **ELFORM 5 = 1-point ALE (one property applied to the ALE area)**
- **ELFORM 6 = 1-point Eulerian (one property applied to the Eulerian domain)**
- **ELFORM 11 = 1-point ALE multi-material element (applies multiple properties to the ALE area)**
- **ELFORM 12 = 1-point ALE single material with void**

In the workshop model, all the properties of water were applied to the ALE domain and the air area was set to Void, so a solid element with ELFORM = 12 was applied to the ALE area.

Define properties of the part by applying the created properties and section as shown below. Water and Air apply similarly and Fuselage selects shell section and rigid property.

- **Keyword Manager → Part → SECID, MID (see image)**



*PART_(TITLE) (3)

1 TITLE

Fuselage

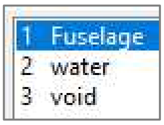
2 PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
1	3	2	0	0	0	0	0

*PART_(TITLE) (3)

1 TITLE

water

2 PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
2	1	1	1	0	0	0	0



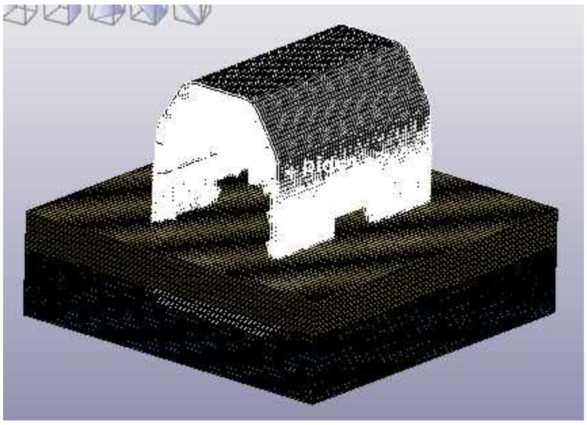
1 Fuselage
2 water
3 void

▲ Define part properties (same for water and air)

Boundary / Loading Condition

In this step, the helicopter's fuselage velocity is set using * Initial_Velocity and the air zone is defined by * Initial_Void_Part.

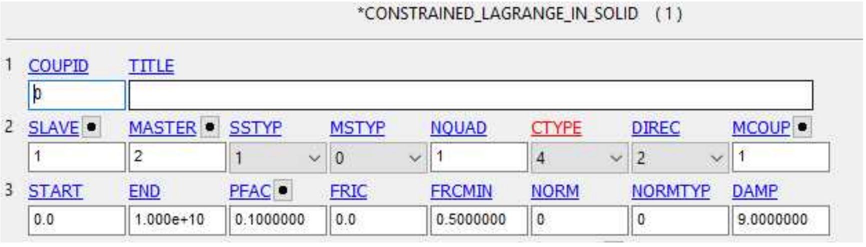
- **Model and Part → Create Entity → Initial Velocity | Cre → Select whole fuselage, Vy=-8m/s → Done**
- **Keyword Manager → Initial | Void_Part → Select Air → Done**



▲ Initial velocity set up

The next step is for Lagrangian fuselage parts to establish contact conditions with the Eulerian surface. In ***Constrained_Lagrange_In Solid**, select the ALE area as the **Master** and then helicopter fuselage as the **Slave**. Set Water and Air to one **Part Set** and define them as shown below. **CTYPE** is an option for contact algorithm and input **CTYPE** = **4** which is the contact condition between Shell and Solid using Penalty Method.

- **Model and Part → Create Entity → Set_Data | Set_Part | Cre → Select all Air and Water to set Single Set → Done**
- **Keyword Manager → Constrained_Lagrange_In Solid → STYPE=1, MTYPE=0, select fuselage for a slave, ALE for a master, CTYPE=4 → Accept**
-

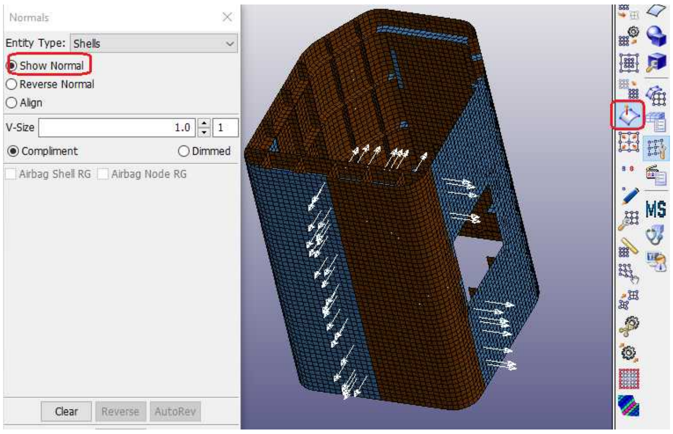


*CONSTRAINED_LAGRANGE_IN_SOLID (1)

1	COUPID	TITLE						
	b							
2	SLAVE ●	MASTER ●	SSTYP	MSTYP	NQUAD	CTYPE	DIREC	MCOUP ●
	1	2	1	0	1	4	2	1
3	START	END	PFAC ●	FRIC	FRCMIN	NORM	NORMTYP	DAMP
	0.0	1.000e+10	0.1000000	0.0	0.5000000	0	0	9.0000000

▲ Define contact condition in *Constrained_ Lagrange_In Solid

One thing to note is that the shell normal direction of the structure in which the collision occurs in the water domain must coincide with the direction of the collision. Use **Normals** → **Shell** to align the normal direction. If **Show Normal** is selected, the shell's normal orientation appears as shown. The parts of the shell that collide with the water domain should be aligned to match this normal direction using **Reverse Normal** or **Align** option.


Normals

Entity Type: Shells

● Show Normal

○ Reverse Normal

○ Align

V-Size 1.0 1

● Compliment ○ Dimmed

☐ Airbag Shell RG ☐ Airbag Node RG

▲ Define Shell Normal Direction

NOTE Types of Fluid-Structure Interaction (Coupling of ALE and Lagrangian)

The FSI, which is the interaction between the fluid region and the structural region, can be classified into three types and the contact condition setting method for the interaction is different. In the exercise, ***Constrained_Lagrange_In Solid** was used because the Lagrangian structure collides with ALE water domain

- **When a Lagrangian structure collides with a Lagrangian object**
  → * CONTACT (How to set the contact condition of a collision between general structures)

- **When an ALE object collides with an ALE object**
  → Merged Nodes (called Advections as the material transfer as the nodes of the boundary overlap each other)

- **When Lagrangian structure collides with an ALE object,**
  → Use * Constrained_Lagrange_In Solid to set the contact condition.

In the *Constrained_Lagrange_In Solid keyword, if ALE and Lagrange areas are defined, LS-DYNA calculates the coupling force for the depth penetrated by the penalty method while searching for the intersection in the analysis process and then read the boundary again as the interpretation proceeds.

Analysis Condition

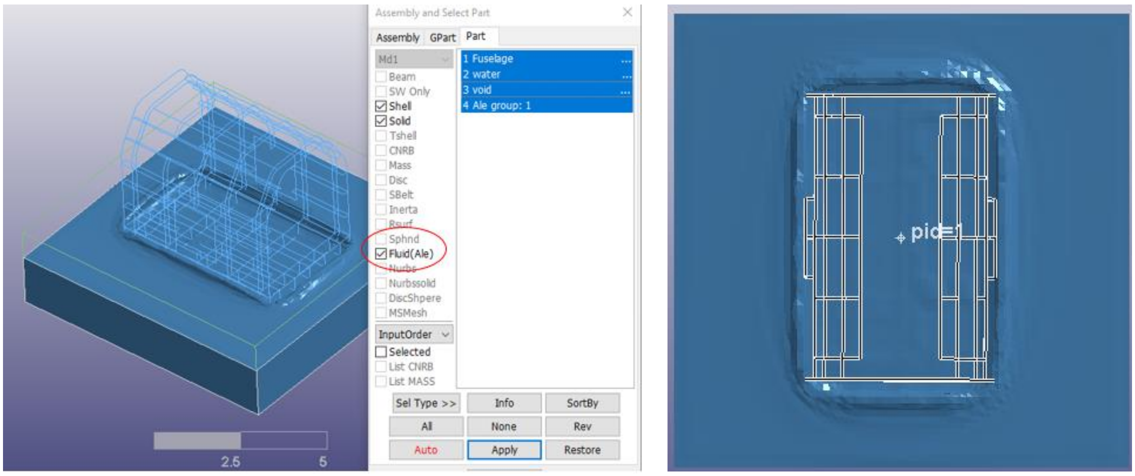
The following steps are done to set the analysis condition. Create a basic control card for ALE and set the analysis end time to 0.2sec. Database FSI is an option to extract data generated through fluid-structure interaction during the embarking process. * **Control_ALE** is a keyword that activates and adjusts the interaction between ALE areas and applies the default value.

- **Control | Termination → ENDTIM = 0.2**
- **Control | ALE → Default**
- **Database→ ASCII_option → DT=0.001, GLSTAT, MATSUM, NCFORC**
- **Database→ FSI → DT =0.001, STYPE=1, Select fuselage → insert**
- **Database→ BINARY_D3PLOT → NPLTC=50**

This completes the modeling of the water landing analysis of the helicopter body.

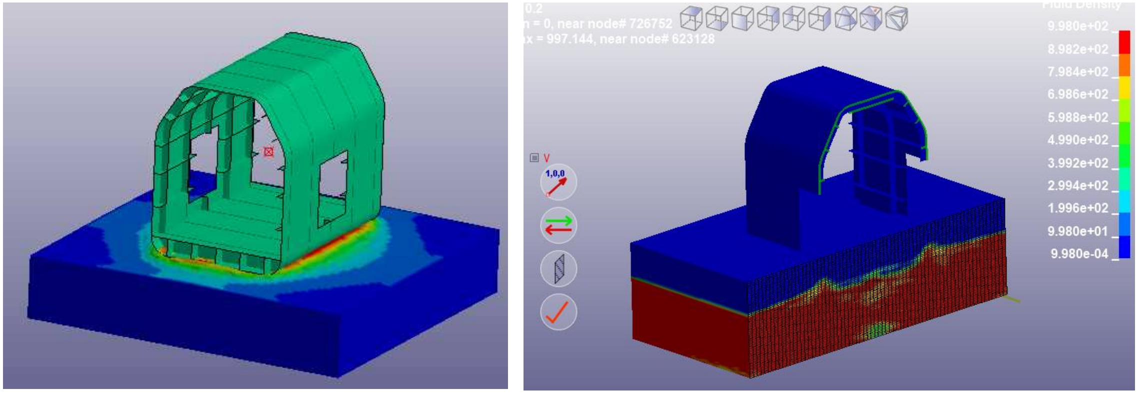
Analysis Result

If you proceed with solving the water landing analysis, with an I7 CPU with 16G Ram PC will take at least 1-2 hours. After the analysis is complete, open the **D3plot** and check it. To check the deformation of water in the ALE domain, select **Assembly and Select Part → Fluid (ALE)** to see the deformation of the water domain.



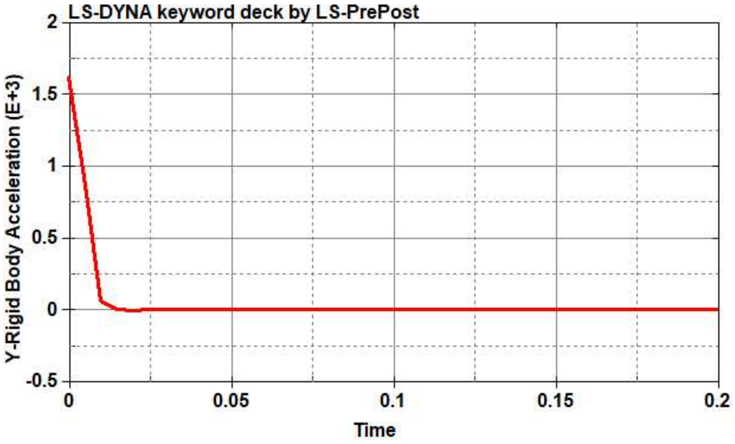
▲ Check the Water Domain deformation area in Post-Processing, select Fluid (ALE)

The figure below shows **Fringe Component → NDV | the result velocity and Misc → Fluid density** are checked through the cross-section. You can check the shape, speed, and density of water in the process.



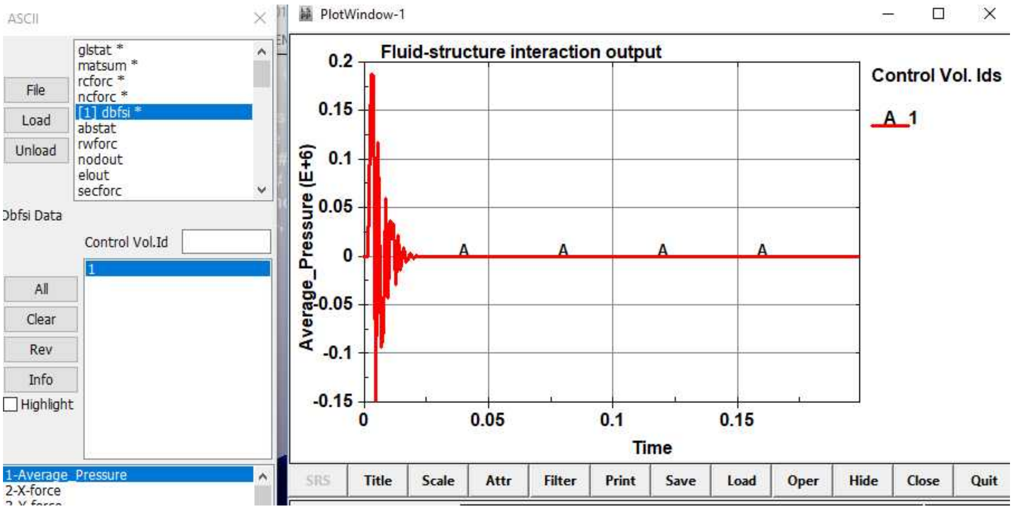
▲ Result Velocity and Fluid Density Distribution in Post-Processing

In addition, the kinetic energy distribution extracted from the Fuselage part can be confirmed as follows. The water impact acceleration curve shows about 1.5kJ which shows a rapid decrease in kinetic energy.



▲ Kinetic Energy Distribution of Helicopter Fuselage Parts

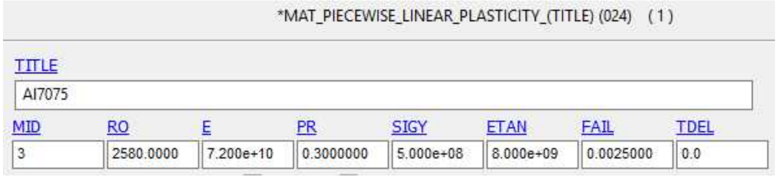
Since the helicopter fuselage is modeled as a rigid body in this exercise, the deformation of the fuselage cannot be confirmed, but the magnitude of the pressure acting on the surface of the fuselage can be extracted through **DBFSI**. In the Post Process, select **ASCII → DBFSI | Load → Average Pressure → Plot** allows you to plot the average pressure at the surface as follows: It can be seen that the average pressure gradually decreases from 180 kPa.



▲ Average pressure distribution extracted from helicopter fuselage landing surface

Reviewing the results of the analysis in the macroscopic aspect, it can be seen that they are somewhat different from the actual phenomena. First of all, the influence of the end face in the water domain should be minimized. In the practice, the repulsive force of the water area is greater than the actual behavior due to the influence of the end face. In this practice workshop, the emphasis was placed on the overall interpretation process and the understanding of the application principle of ALE rather than worry on scientific analysis accuracy. Additional analysis is recommended to perform by increasing the ALE domain area more than two to three times.

In addition, since it was modeled as a rigid body, the surface pressure due to the impact load could be calculated, but the deformation and damage of the fuselage could not be obtained. So, let's import the provided model W12_Water_ALE_Elastic.k in LS-Prepost. The only difference from the previous model to this is that the size of the entire model is reduced to only a partial frame area and the physical property model is changed to * **MAT024_Piecewise _Linear_Plasticity** (from Rigid to aluminum properties). Since the ALE area is very small, we adjusted some water properties to eliminate negative advection errors. Proceed with the analysis and examine the results. It can be seen that von mises stress occurs in the helicopter fuselage structure having elastic characteristics so that deformation and breakage occur. In order to identify the weak spots at the onset load of the structure, elastic properties should be applied. In the case of the flexible analysis of the deformable structure and the fluid, it will take considerable analysis time.



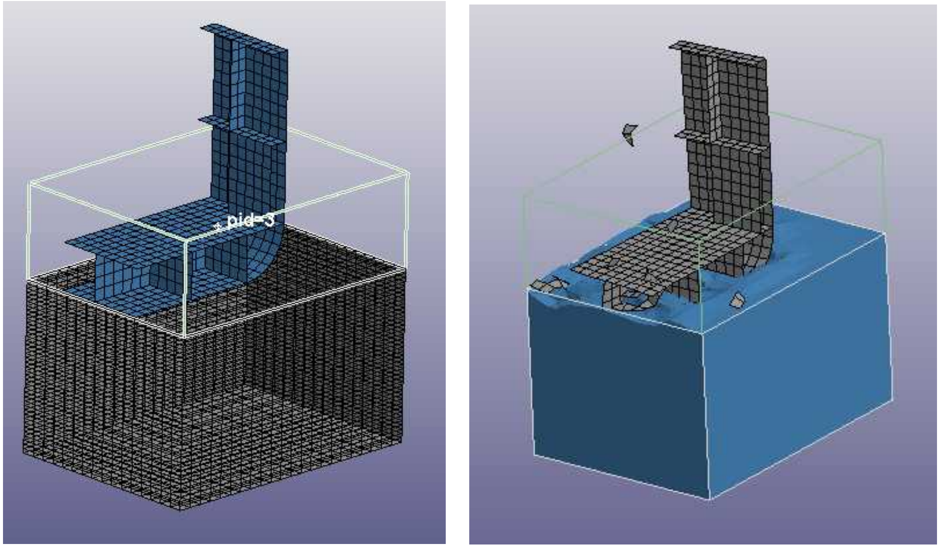
*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE

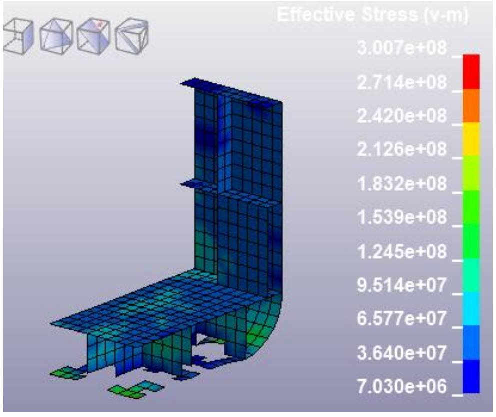
Al7075

MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
3	2580.0000	7.200e+10	0.3000000	5.000e+08	8.000e+09	0.0025000	0.0

▲ Aluminum property of helicopter body



▲ Analysis Model and water landing analysis for partial frames with elastic properties



▲ Element deletion due to excessive Von mises stress

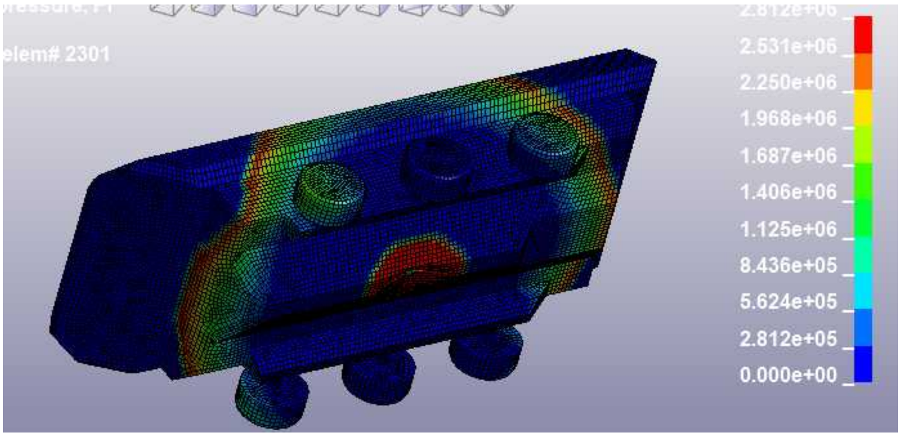
So far, we have analyzed the water landing analysis of the helicopter body using the ALE technique. This was a practice model used to learn the ALE analysis technique and I hope that you have some understanding of ALE's application concept. The ALE analysis technique is a very effective analysis technique that can be used in many industries where large deformation occurs. For example, this technique can be useful in various fields such as a collision of ductile materials such as bird strike or dropping of a fuel tank, analysis of plastic processing phenomena such as forging or stamping and also a collision of fluid and structure such as sloshing of liquids. I hope you understand this ALE scheme and apply it appropriately.

NOTE Negative Volume Error

Perhaps a minor mistake in the modeling process will result in a Negative Volume Error message. If the collision part is subjected to very severe deformation, this error message occurs, when the volume is calculated as a negative value. This error often occurs when Failure Criterion is not set. This error is inevitable if the mesh size or the remeshing or smoothing technique is not applied. So how do we resolve this error? First of all, it is most effective to set the failure criterion, that is, the option of excluding the element from the calculation process when a failure threshold is reached by setting the failure criterion.

* Set ERODE = 1 in **Control_Timestep** to delete the corresponding element of Negative volume. Also, if you enter a non-zero value for DTMIN in * **Control_Termination**, in most cases, the element corresponding to the negative volume is deleted and the analysis continues. Another useful tip is to apply Hourglass option 4 or 5 when using a solid element formulation to make the element stiffen in large deformations, slowing down the creation of negative volumes. Of course, even with Erode and DTMIN, negative volume errors can occur for many different reasons. Hourglass will be discussed in detail later, but for low-speed collision analysis (~ several cm / s) it is better to use type 6 hourglass and for high-speed collision analysis (~ several m / s) use type 2 and 3 to reduce the possibility of negative volume error occurrence.

Chapter 13. Explosive Analysis



Objectives

- ✓ Learning the concept of explosion analysis
- ✓ Practicing the procedure of *Load_Blast_Enhanced keyword
- ✓ Demonstrating explosion load analysis results

Duration: 60 min

Application: Defense, Blast Loading, Vehicle

Model files

- W13_Vehicle_Mesh.bdf: Vehicle mesh model (Nastran)
- W13_INITIAL_Vehicle_Mesh.k: Initial state model
- W13_SETUP_BlastVehicle.k: Final state model

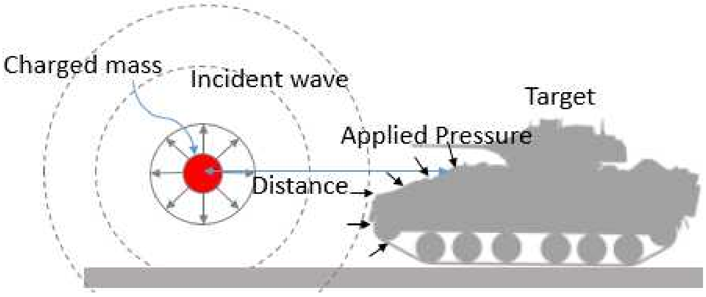
Introduction of Explosive Blast

Due to the recent increase in terrorist activity, threats havocked by terrorists using explosives on vehicles, aircraft, buildings, etc. are gradually increasing. One of the important functions of LS-DYNA is the simulation of such explosives. Explosion analysis using LS-DYNA is widely used in the design of many military vehicles and other military installation facilities and it is also a hot topic of analysis field in many user-conference. In fact, many accidents have occurred in the war against Iraq, the war against Afghanistan and the IS sweeping operation.

▲ Ground Explosion Test on Armored Vehicles

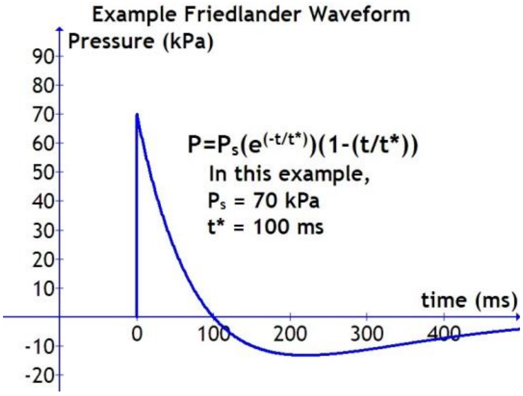
LS-DYNA provides three analysis methods for blast loading analysis. The method is based on an empirical method called ***Load_Bast_Enhanced (LBE),** which uses the **CONWEP** explosive analysis method described in the US Army Handbook. The other method is to apply **Arbitrary Lagrangian-Eulerian (ALE).** The other method is the **Multi-Material ALE** which combines the above two and the other is ***Particle_Blast** method using **SPH**. In this chapter, we will practice an analytical model in which the mines explode on the underside of armored vehicles using the ***Load_Blast_Enhanced** method, which is relatively easy to apply.

First, a physical understanding of the phenomena is necessary for the interpretation of the explosion. When explosives explode in the atmosphere, tremendous energy is transmitted in the form of pressure to the target through the surrounding air as an incident shock wave. The transmitted shock wave is converted into a reflected shock wave and the definition of this waveform is set to Blast Loading. * **Load_Bast_Enhanced** keyword can be applied to Lagrangian structure directly by simply setting shock waves using weight, distance, and position of explosive.



▲ Blast Loading Concept

Reflected Incident Wave has several theories, which are defined by Friedlander's empirical formula, the simplest but most practical. **Load_Blast_Enhanced** can simply simulate the explosion in the atmosphere for spherical TNT explosives in general.



$$P(t) = P_s e^{-\frac{t}{t^*}} \left(1 - \frac{t}{t^*}\right)$$

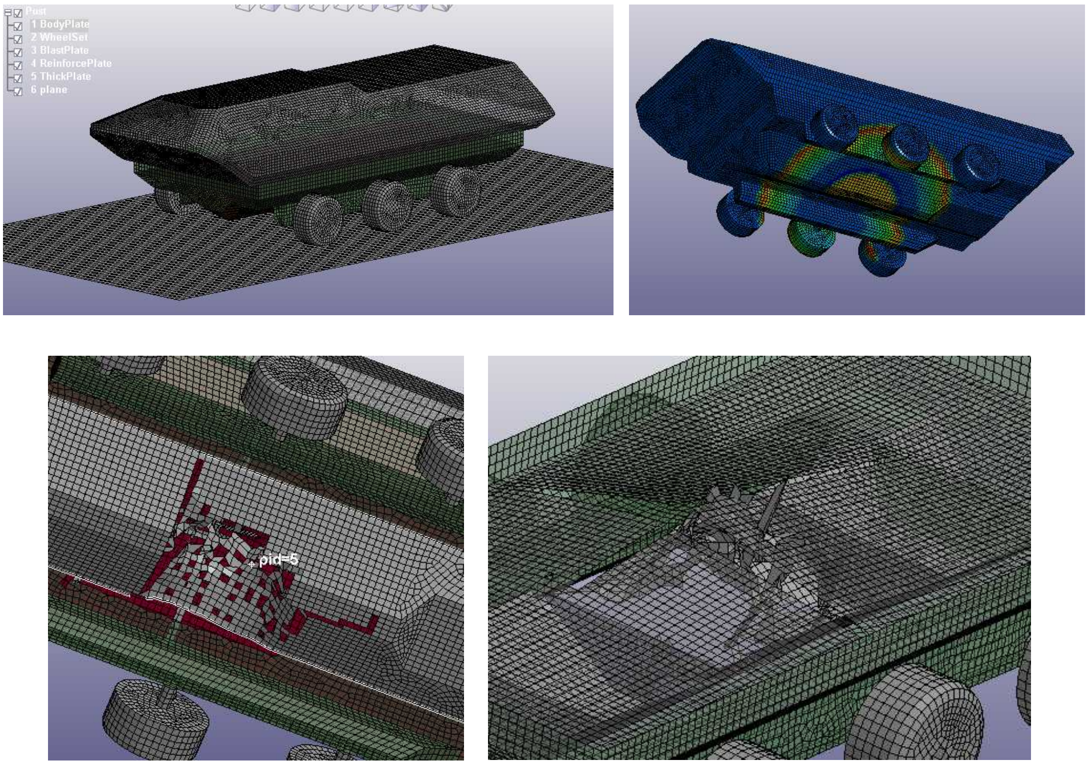
▲ Graph of time with Reflected Incident Wave

W13

Workshop 13.
Ground Explosion Analysis for Vehicles

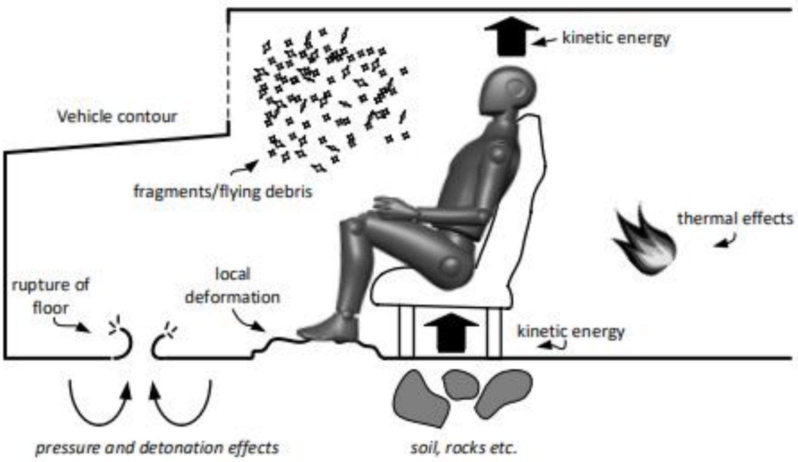
Introduction

In the explosive analysis of the vehicle, a simplified armored vehicle model is used. It is a vehicle made of the high strength steel plate and the simplified model is composed of the shell elements. In fact, the model is similar in size to the vehicle used on the battlefield. During the actual operation, the explosion on the rear side caused a lot of damage to the vehicle as well as the occupants. Therefore, the research on the explosion protection design of the rear part is being actively conducted. According to the research results, backplates of various types (bubbles, wedges, sandwiches, trusses, etc.) are designed, because the greatest damage occurs when the blast shock wave is transmitted perpendicular to the plate. This model is an armored vehicle equipped with a wedge-type bottom protection plate.



▲ Mine explosion model for armored vehicles

The most common cause of the explosive threat of armored vehicles is the self-made explosive IED. When an IED explodes on the bottom of the vehicle, a severe deformation or hole is made in the floor plate and the shock wave propagates inside, causing severe damage and also injuring the occupant by splashing the debris. In addition to this, a sudden movement of the vehicle may cause serious damage such as a collision to nearby vehicles or ignition of internal flame from the fuel. Shock wave pressure that can damage the human body is known to cause irreversible damage to the lungs when the pressure is 200kPa or more. In general, the shock wave of a single grenade explosion is known to be 60 ~ 70kPa. The key to the design of the protective plate is to absorb and disperse the explosion pressure as much as possible so as not to penetrate the inner vehicle hull from the expected explosion threat.



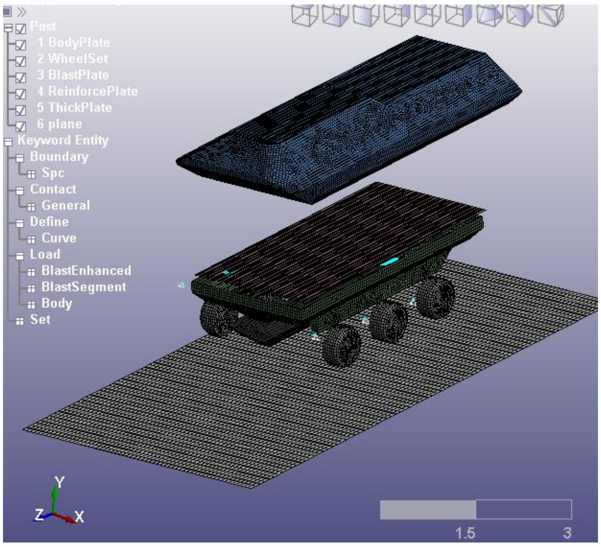
▲ Concept of a vehicle and occupant damage due to IED (Stankiewicz et al. 2015)

In this exercise, the explosion analysis was performed assuming that about 50kg of antitank mine TNT explosives that would explode on the ground where the armored vehicle is located. The unit system is applied to the meter system considering the size of the vehicle.

- **Unit: m, sec, kg, N, Pa**

Import Mesh

Import the model '***W13_mesh.k'***, here the mesh is already created for the explosion analysis. The model is created by CAD program including the mesh and then imported as a Nastran model (* .bdf). If you import '***W13_Initial_Vehicle_mesh.k***', a mesh model consisting of six parts will appear as shown below. If you import the attached ***W13_VehicleMesh.bdf*** model, a single part model is generated. Then, segregate this model into six parts as shown below image.

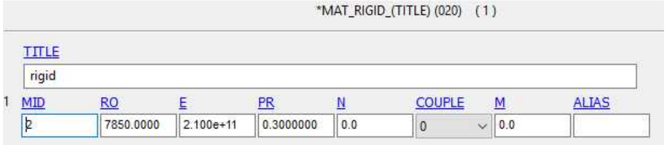


▲ 6-part vehicle model (Explode view)

Material Property

The material of the vehicle is assigned using **MAT003_Plastic_Kinematic** properties containing high strength steel data. Define **FS** (Fail Strain) = 0.0015, which enables the element to be removed when the deformation reaches Fs by compression wave. Because it is a simplified model, it was applied to the body as a single material and only the bottom plate was applied with increased thickness (80mm). Ground plane applied **Rigid** properties.

- **Keyword Manager → MAT | 020-Rigid → steel material properties for plane**
- **Keyword Manager → MAT | 003-Plastic_Kinematic →steel material properties for structure**



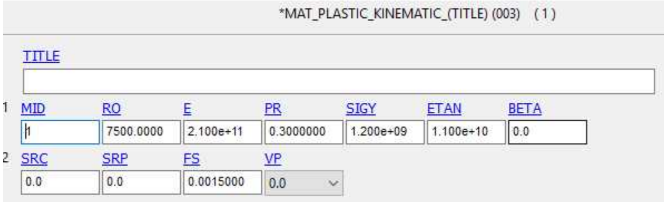
*MAT_RIGID_(TITLE) (020) (1)

TITLE

rigid

MID	RO	E	PR	N	COUPLE	M	ALIAS
2	7850.0000	2.100e+11	0.3000000	0.0	0	0.0	

▲ MAT020_Rigid Properties



*MAT_PLASTIC_KINEMATIC_(TITLE) (003) (1)

TITLE

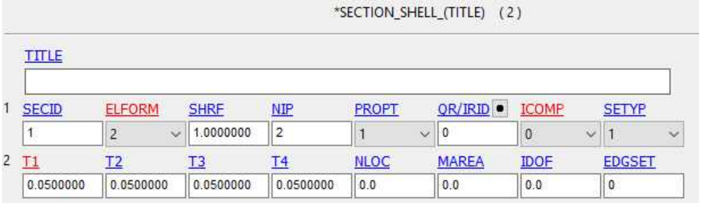
1	MID	RO	E	PR	SIGY	ETAN	BETA
	1	7500.0000	2.100e+11	0.3000000	1.200e+09	1.100e+10	0.0
2	SRC	SRP	FS	VP			
	0.0	0.0	0.0015000	0.0			

▲ MAT003_Plastic Kinematics Properties

Element Property

After defining the properties, the properties should be assigned to the target part accordingly. First, define the section characteristics of the vehicle composed of shell elements. Enter **ELFORM** = 2. Two sections are created: the vehicle chassis thickness is 50mm and the bottom plate is 80mm.

- **Keyword Manager → Section_Shell → ELFORM=2, create 2 sections (thickness 50mm, 80mm)**



*SECTION_SHELL_(TITLE) (2)

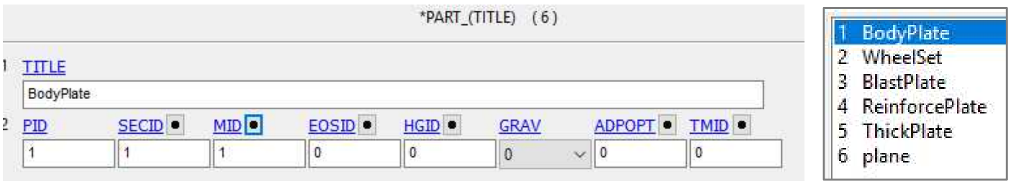
TITLE

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
	1	2	1.0000000	2	1	0	0	1
2	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
	0.0500000	0.0500000	0.0500000	0.0500000	0.0	0.0	0.0	0

▲ Shell section properties for vehicle chassis (50mm)

Update the part properties by applying the created material properties and section as shown below. Since this is a simplified model, set the thickness as 50mm section for all vehicle parts except for the thick plate part and set the 80mm part only for the thick plate part. Ground planes define **Rigid** properties, and all parts of the vehicle are defined using **MAT003** properties.

- **• Keyword Manager → Part → SECID, MID (see image)**



*PART_(TITLE) (6)

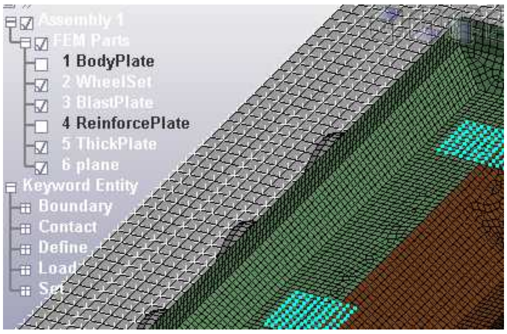
1	TITLE							
	BodyPlate							
2	PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
	1	1	1	0	0	0	0	0

1 BodyPlate
2 WheelSet
3 BlastPlate
4 ReinforcePlate
5 ThickPlate
6 plane

▲ Part setup

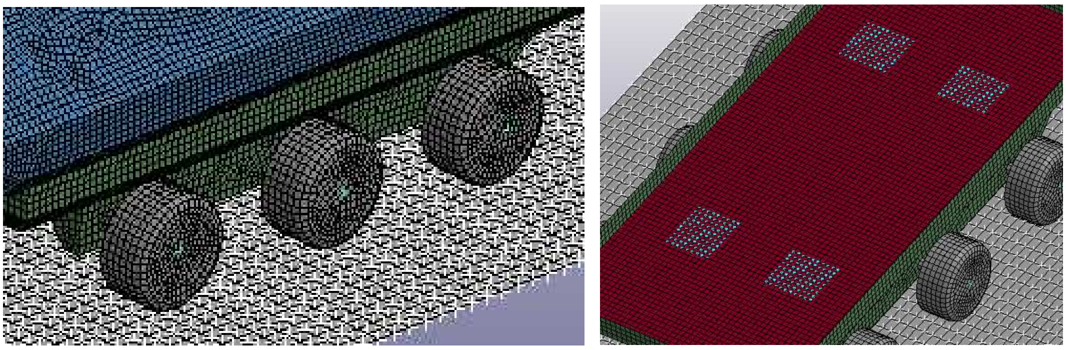
Mass / Element Generation

Since the weight of armored vehicles is one of the main factors affecting the explosion analysis, it creates a dummy mass element to make the weight of the vehicle more for a realistic approach to the physical model. First, set the weight of 50kg on the wheel and set the specific area of 2kg per node inside the vehicle to simulate the weight of the components inside the vehicle. When dealing with a model consisting of multiple parts like in this exercise model, it is easy to model internal components by expanding the model tree on the left of the viewport and using it as shown below.



▲ Model tree structure

- **Model and Part → Create Entity → Element | Mass → Cre | Mass = 50kg → Select nodes at the center of the wheel(6) → Apply**
- **• Model and Part → Create Entity → Element | Mass → Cre | Mass = 2kg → select small are in the Reinforce plate (4) → Apply**

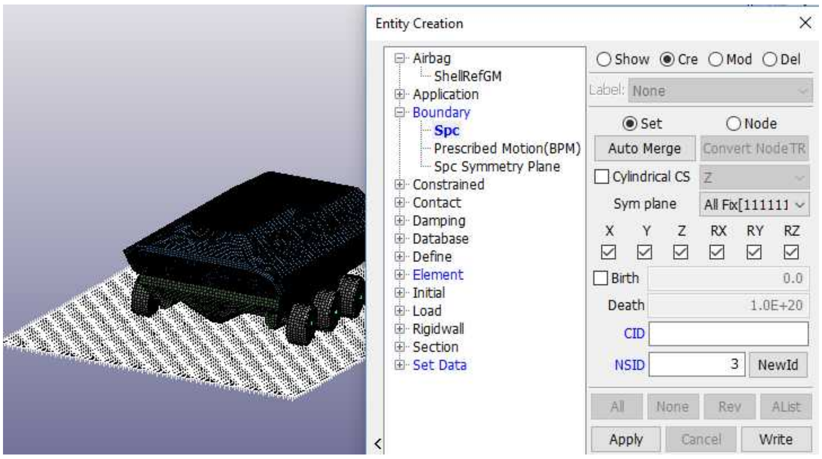


▲ kg mass element (wheel), 2 kg mass element (internal gusset)

Boundary Condition

The ground plane is fully restrained as shown in the figure.

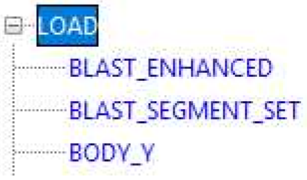
- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Area → Select plane and check DOF → Apply**



Sym plane All Fix[111111]
X ☑ Y ☑ Z ☑ RX ☑ RY ☑ RZ ☑
☐ Birth 0.0
Death 1.0E+20
CID
NSID 3 NewId

▲ Boundary condition setting screen

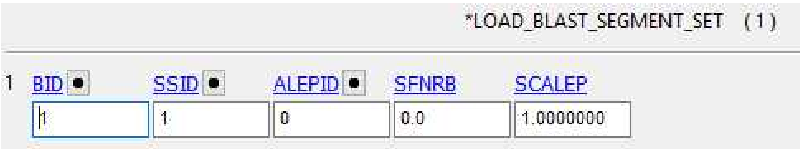
After creating the boundary conditions, the most important load conditions are set. The load condition applied in this exercise uses three keywords as shown below. We use load Body_Y to the weight of the vehicle, Blast_Enhanced to define the magnitude and location of the explosion load and Blast_segment_set keyword to define the target of the explosive as a segment.



▲ Load Keyword

The ***Load_Blast_Enhanced** keyword sets the weight of the explosive (TNT), M = 50, the location of the center point of the explosive (2.5, 0, 5), the explosive definition unit system (2 = default, the SI [meter] unit system) and the type of explosion. If you want to use **UNIT** = 3 (English unit), you need to input **CFM**, **CFL**, etc. in 2 columns to convert the data. **Blast** = 1 means explosion on the ground surface and **Blast** = 2 means explosion in the air. In the case of an explosion on the ground, it is not necessary to define a direction of explosion because there is a difference in size considering the reflected wave. In this case, **BID** means the ID of the segment set that defines the part which the explosion load acts on. Therefore, set the entire vehicle as one segment set using ***Set_segment.** Set the segment set like this in ***Load_Blast_Segment_set.**

- **Model and Part → Create Entity → Set Data | Segment → Area → select all vehicle → Apply**
- **Keyword Manager → Load_Blast_Segment Set → select generated segment set**

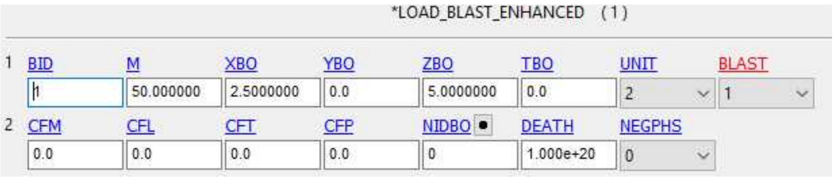


*LOAD_BLAST_SEGMENT_SET (1)

1	BID ●	SSID ●	ALEPID ●	SFNRB	SCALEP
	1	1	0	0.0	1.0000000

▲ Load_Blast_Segment setup

- **Keyword Manager → Load_Blast_Enhanced → input values(see image)**



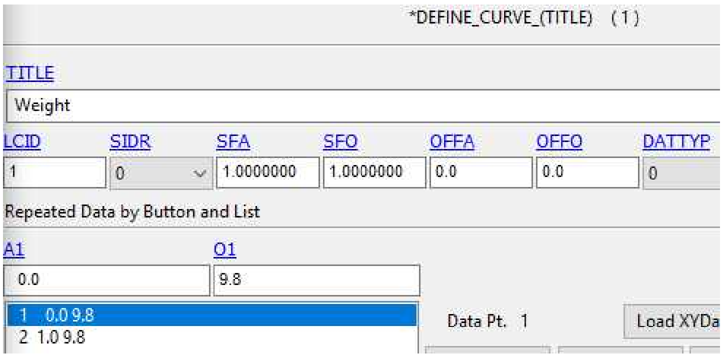
*LOAD_BLAST_ENHANCED (1)

1	BID	M	XBO	YBO	ZBO	TBO	UNIT	BLAST
	1	50.000000	2.5000000	0.0	5.0000000	0.0	2	1

2	CFM	CFL	CFT	CFP	NIDBO ●	DEATH	NEGPHS
	0.0	0.0	0.0	0.0	0	1.000e+20	0

▲ Load_Blast_Enhanced setup

To set the gravity load on the vehicle, i.e. the magnitude of the acceleration due to gravity is created through ***Define_Curve**. Create the defined curve so as to be uniformly 9.8m / s during the analysis time, select **LCID** created in * **Load_Body_Y** (consider the vehicle alignment direction) and enter the gravity direction **SF = -1**.



*DEFINE_CURVE_(TITLE) (1)

TITLE: Weight

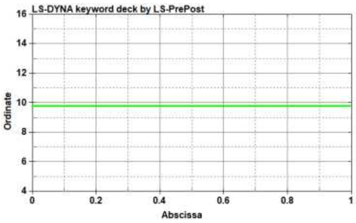
LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP
1	0	1.0000000	1.0000000	0.0	0.0	0

Repeated Data by Button and List

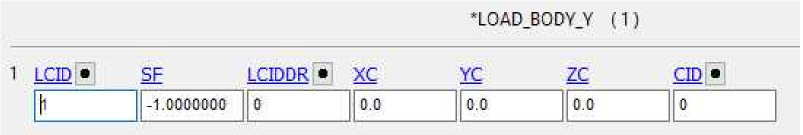
A1	O1
0.0	9.8

1 0.0 9.8
2 1.0 9.8

Data Pt. 1



- **Keyword Manager → Load_Body_Y → input curve ID and SF (see image)**



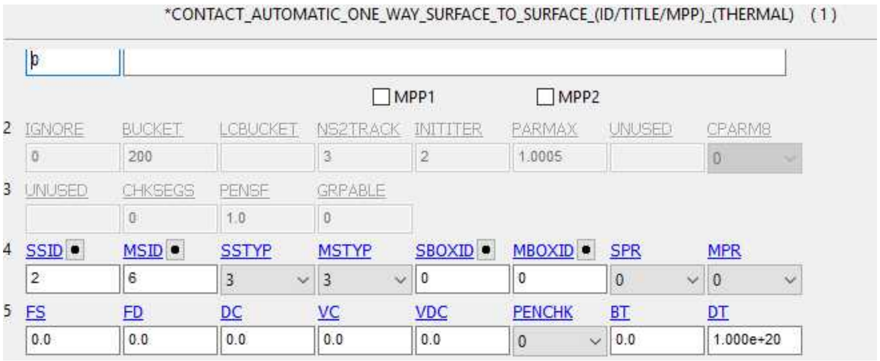
*LOAD_BODY_Y (1)

1	LCID ●	SF	LCIDDR ●	XC	YC	ZC	CID ●
	1	-1.0000000	0	0.0	0.0	0.0	0

▲ Load_Body_Y setup

Contact Condition

The contact condition is defined using * **Contact_Automatic_ Oneway_Surface to Surface**, which is a simple contact condition between the wheel part of the vehicle and the ground plane.



*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (1)

0

☐ MPP1 ☐ MPP2

2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	2	6	3	3	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.0	0.0	0.0	0.0	0.0	0	0.0	1.000e+20

▲ Contact Condition setup

Analysis Condition

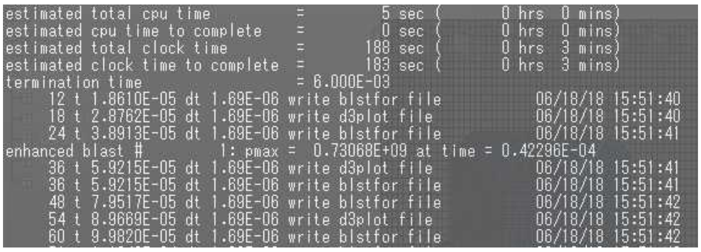
The following step is to set the analysis condition. The analysis end time is set to 0.006 sec in consideration of the time when the shock wave is transmitted to the bottom plate and propagated throughout the vehicle.

- **Control | Termination → ENDTIM = 0.006**
- **Database→ ASCII_option → DT=2e-5, GLSTAT, MATSUM, NCFORC**
- **Database→ BINARY_BLSTFOR → DT =2e-5**
- **Database→ BINARY_D3PLOT → NPLTC=200**

After this step, the explosion analysis model for the vehicle has been completed.

Analysis Result

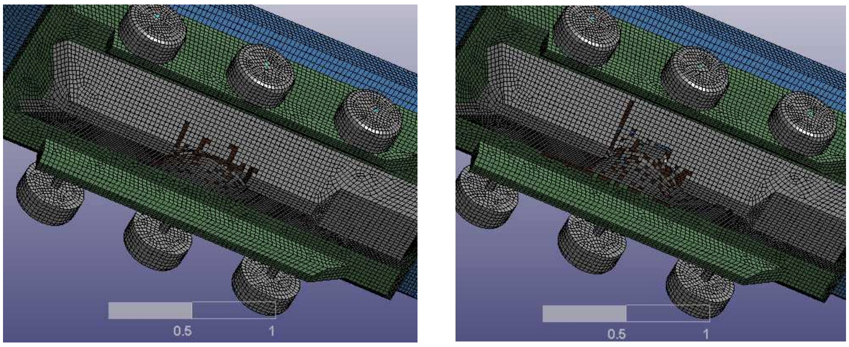
If you proceed with the analysis by submitting the calculation on a good PC (I7 CPU, 16G ram), it will be completed within 20-30 minutes. If you check the log screen during the analysis, you can see that the enhanced blast max pressure 0.73GPa occurred in 0.4e-4 seconds when the shock wave reached the model. You can also see how elements fail. Typically, a shock wave of about 10 kPa breaks the window and causes the deformation of the metal. Considering that the shock wave is about 100 kPa, it can be seen that a huge amount of explosion pressure has been generated considering that the pressure is enough to puncture the metal.



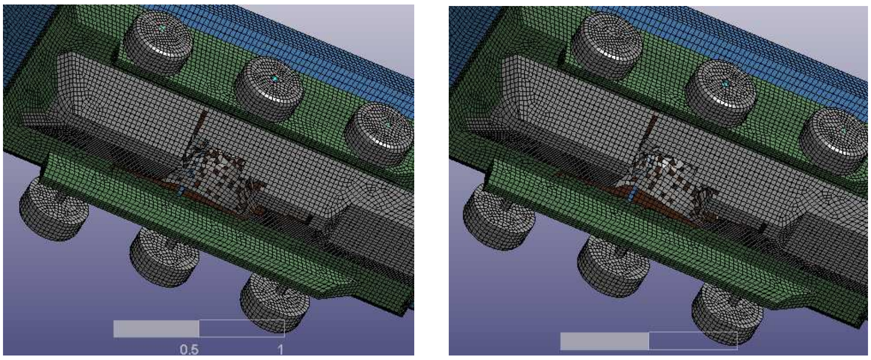
```
estimated total cpu time              =         5 sec (        0 hrs  0 mins)
estimated cpu time to complete        =         0 sec (        0 hrs  0 mins)
estimated total clock time            =       188 sec (        0 hrs  3 mins)
estimated clock time to complete      =       183 sec (        0 hrs  3 mins)
termination time                      = 6.000E-03
     12 t 1.8610E-05 dt 1.69E-06 write blstfor file           06/18/18 15:51:40
     18 t 2.8762E-05 dt 1.69E-06 write d3plot file            06/18/18 15:51:40
     24 t 3.8913E-05 dt 1.69E-06 write blstfor file           06/18/18 15:51:41
enhanced blast #         1: pmax =  0.73068E+09 at time = 0.42296E-04
     36 t 5.9215E-05 dt 1.69E-06 write d3plot file            06/18/18 15:51:41
     36 t 5.9215E-05 dt 1.69E-06 write blstfor file           06/18/18 15:51:41
     48 t 7.9517E-05 dt 1.69E-06 write blstfor file           06/18/18 15:51:42
     54 t 8.9669E-05 dt 1.69E-06 write d3plot file            06/18/18 15:51:42
     60 t 9.9820E-05 dt 1.69E-06 write blstfor file           06/18/18 15:51:42
```

▲ Analysis Progress Log Screen

After the analysis is complete, open the D3plot to check the bottom plate for damage. As shown in the figure below, the 80mm steel plate directly above the explosives is completely destroyed. You can see that the inner plate is completely broken in a cross shape between the weights.

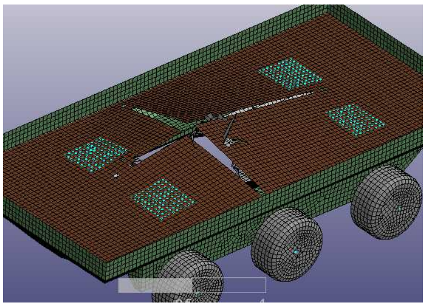


▲ 0.001sec ▲ 0.002sec



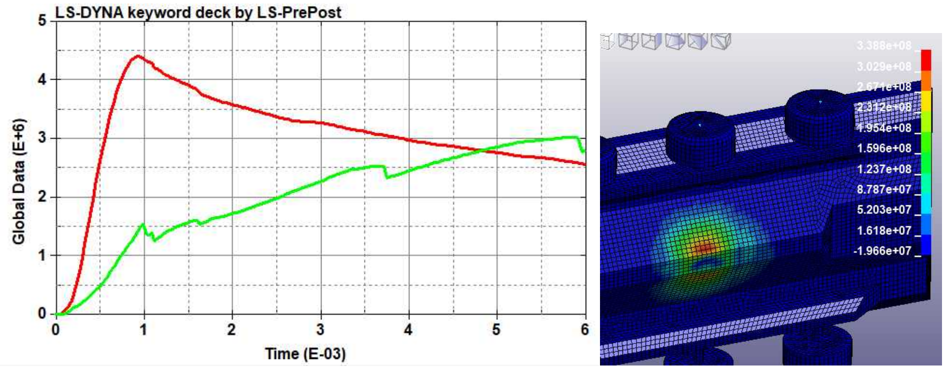
▲ 0.004sec ▲ 0.006sec

▲ Damage Progress of bottom Plate by Explosion



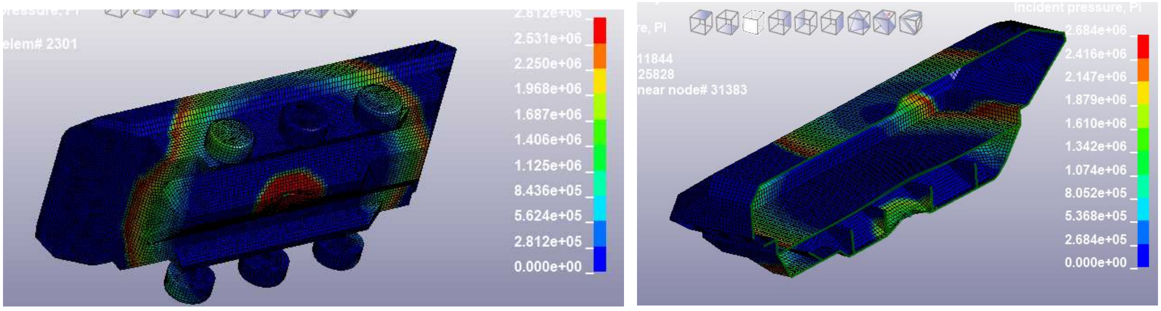
▲ Damage Progress of Internal Plate by Explosion

The figure below shows the kinetic energy (red), internal energy (green) and pressure distribution fringe pattern throughout the vehicle system.



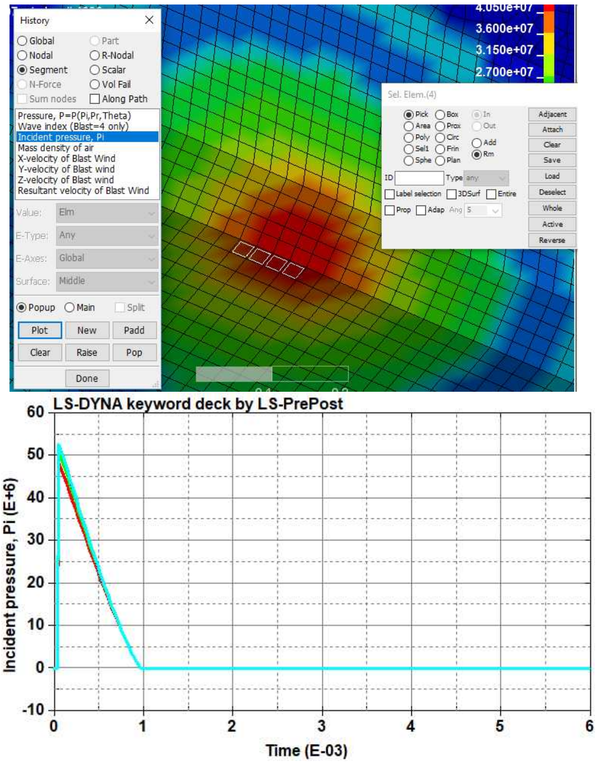
▲ Kinetic / Internal Energy Distribution and Pressure Distribution

Let's open the **BLSTFOR** file. The **BLSTFOR** file is the result file that stores the data related to the explosion. Only the result of the part set as **Blast_Segment** in the model is saved. Key data includes wave index, pressure, incident pressure, the mass density of air and blast wind velocity. The figure below shows the result of the incident pressure propagating through the car body.



▲ Kinetic / Internal Energy Distribution and Pressure Distribution

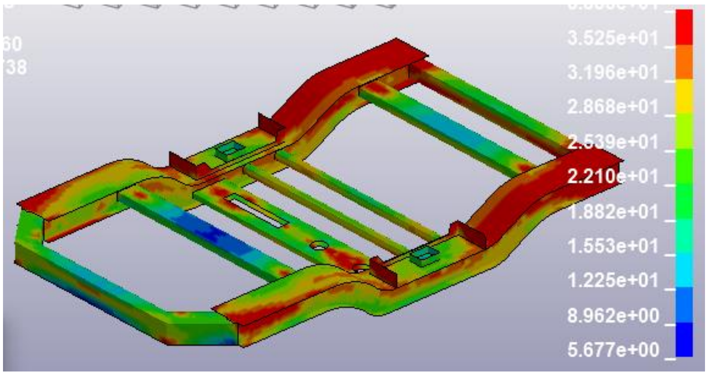
In the **BLSTFOR** output file, select **History** and plot some of the elements in the area where the shock wave is transmitted as shown below. We can see that the distribution of shock waves over time is almost similar to that of Friedlander, which was introduced earlier in this chapter. **DT** is set to 2e-5, but much smaller than this, you will see a more accurate distribution.



▲ Incident pressure distribution over time

This workshop model demonstrated the basic procedure of analyzing explosives using ***Load_Blast_Enhanced**. Experiments on structures for explosives are quite limited in many respects and this numerical approach is widely used in the design of the structure. However, because the phenomenon of the explosion itself has considerable nonlinearity, in order to improve the reliability of the analysis results, it is necessary to correct various variables and compare them with the available experimental data. Recent studies show that explosive analysis using **Multi-Material ALE (MMA)** or **SPH** method in order to get more realistic results. I hope this course has helped you understand the basic explosive analysis. Readers who are interested in the explosive analysis are advised to review the experimental data because a significant amount of research is available online.

Chapter 14. Random Vibration Analysis of Bogie Frame



Objectives

- ✓ Learn the concept of fatigue analysis and fatigue properties
- ✓ Modeling Random Vibration of Structures
- ✓ Modeling railway Bogie structures
- ✓ Understand the fatigue life analysis

Duration: 60 min

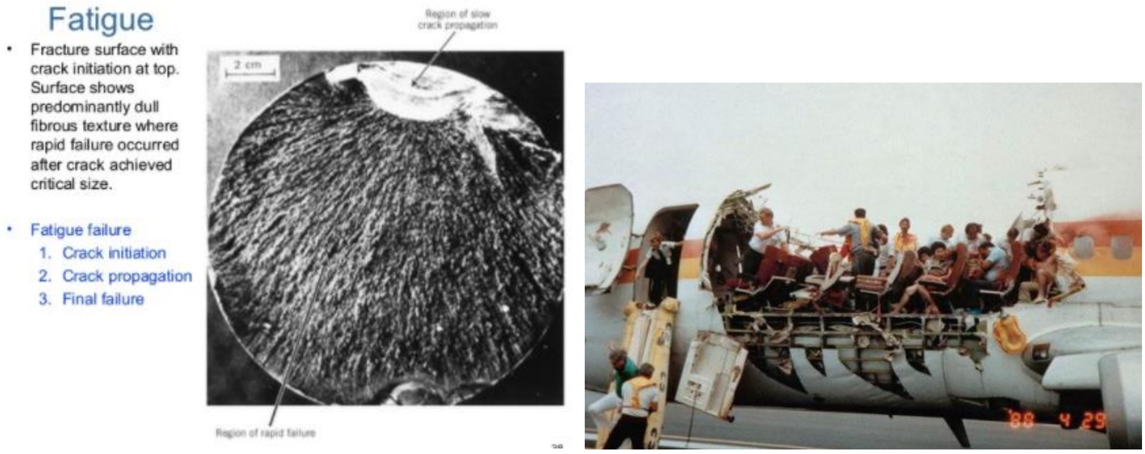
Application: Vehicle, Random Vibration, Fatigue Analysis

Model files

- W14_INITIAL_Bogie_Mesh.k: Initial state model
- W14_SETUP_Bogie_Random_Fatigue_R8.k: Final state model

Fatigue

Fatigue damage is one of the most significant damage mechanisms for any mechanical or electronic parts, but it is still designed without understanding the concept of fatigue damage in mechanical design. Fatigue is a mechanism in which tiny damage accumulates due to repetitive loads, which causes cracks and these fatigue cracks propagate to reach critical points and finally the breaking parts. In the aviation field, the concept of **Durability and Damage Tolerance Analysis (DaDTA)** is applied to the design of the structures. However, more than 70% of defects and accidents in aviation components are caused by fatigue cracking. On the other hand, ground transportation such as cars, trains, ships, etc., or buildings, etc. do not adequately reflect the concept of fatigue cracks in the design and often big accidents occur. The figure below shows a typical fatigue crack cross-section and also a picture of an Aloha 243 aircraft accident that undergone a typical fatigue failure.



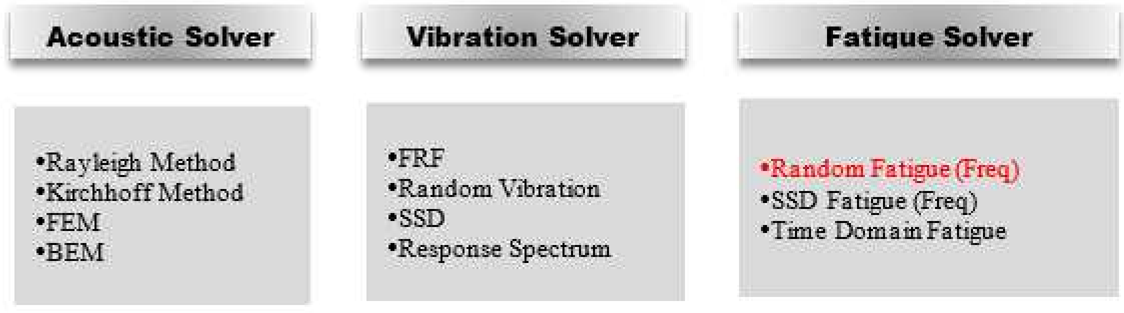
Fatigue

- Fracture surface with crack initiation at top. Surface shows predominantly dull fibrous texture where rapid failure occurred after crack achieved critical size.
- Fatigue failure
  1. Crack initiation
  2. Crack propagation
  3. Final failure

▲ Fatigue Crack Cross Section and Fatigue Damage Case

LS-Dyna has been providing fatigue analysis since the R7 version. Fatigue analysis can be performed in two domains, the time domain, and the frequency domain. In the frequency domain, it can be analyzed in two ways: random vibration and harmonic vibration. Since the theory of fatigue analysis is very vast for one-semester class, the theory cannot be described in this chapter, so only simple concepts necessary for interpretation are explained.

Currently, LS-DYNA is the most used in the automotive industry and the analysis areas required for the design of automobiles include fields such as crashworthiness, occupant safety, NVH, and durability. The vibration and durability analysis functions have been supported in DYNA. The chart below shows the NVH functions provided by DYNA and the area covered in this exercise is random vibration fatigue analysis in the frequency domain.



In general, fatigue analysis is stress-based and strain-based. The optimal approach should be selected in consideration of the operating environment of the structure and the tendency of the working load. In this exercise, we will apply the stress-based fatigue analysis to practice the random vibration fatigue analysis on the Bogie structure, which is the main structure of railway vehicles.

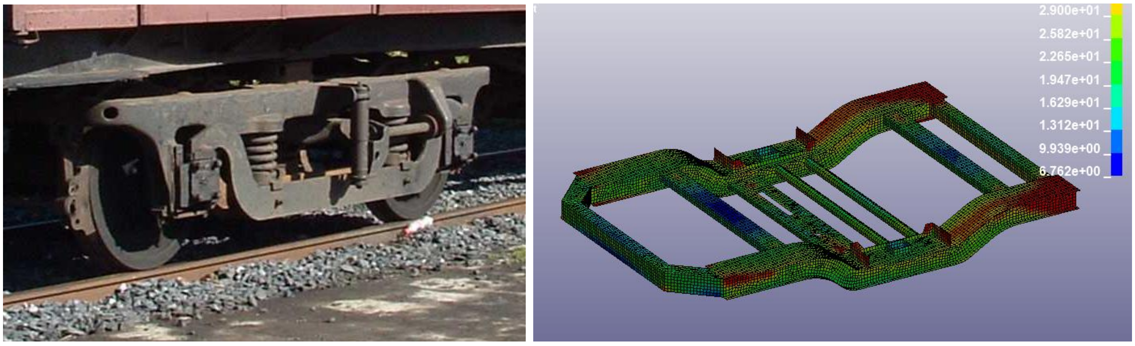
W14

Workshop 14.

Random Vibration Fatigue Analysis of Bogie

Introduction

The bogie of the train is an important component frame structure that connects the carriage and the rails on the ground. The bogie will also be equipped with the wheels of the train and carries the load on the carriages. It is a structure that requires fatigue analysis because it needs to operate on a non-uniform train rail for a long time. In particular, the fatigue due to the standardized load spectrum should be considered, but the random fatigue analysis caused by the vibrations from the structural characteristics of the rail and various forms of the ground must also be performed.



▲ Typical Train Bogie System and Practice Bogie Model

Resonance of structures caused by the influence of external excitation is a major factor that causes the structure to break regardless of the magnitude in the static load regime. Such fatigue analysis methods considering dynamic loads include transient response analysis, vibration fatigue analysis, modal superposition method and sometimes the combination of these. In this exercise, we will practice the random vibration fatigue analysis in the frequency domain to train view structure. The bogie structure of railway vehicles is designed to maintain static and fatigue strengths according to international standards and also to maintain natural frequencies above a certain frequency to avoid resonance. The vibration durability analysis considering the resonance frequency will be performed to calculate the durability life.

.

NOTE Why do we need random vibration durability analysis?

Loads from operating environments in various engineering fields such as railroads, aircraft, ships, buildings, various electronic and mechanical components are almost impossible to predict accurately. In recent years, many sensor technologies have evolved to make it possible to identify an operating environment that has relative effects on the durability of a structure, but there are still many limitations in precise measurement and analysis of vibration environments. In particular, the analysis in the frequency domain is much more difficult.

For example, huge wind turbine blades, various control surfaces of aircraft exposed to turbulence, turbine engine exhaust vents, offshore structures, rough road vehicles, and high speed along uneven railroad tracks. In many cases, such as trains and structures exposed to earthquakes, it is difficult to accurately measure vibration by frequency domain.

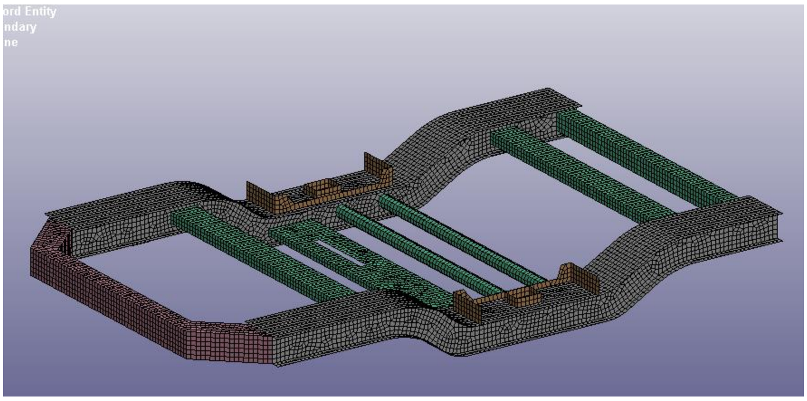
Although accurate life prediction in these structures is the most important factor in securing safety, we have been informed through many news cases that it is overlooked and leads to major accidents. In this respect, it can be said that random vibration durability analysis is important.

▲ Italian bridge accident (2018.8)

Importing Mesh

Import the model ***W14_Initial_Bogie_mesh.k*** from which the mesh is already generated for the railroad Bogie analysis. The model created with four components (Main frame, cross beam, front beam, bracket) will be displayed as shown below. Bogie structures are all made of shell elements and the connection between parts is made based on welding and in the model, nodes are created to share nodes with other parts that help to maintain the model connectivity. Basically, it is important to prove the durability satisfaction in the design process because the fatigue life prediction of the Bogie structure of railroad cars can ensure safety for passengers by securing safety and reliability. The unit system of the analysis model is as follows. Care must be taken while setting the frequency domain or entering the property values.

- **Unit: mm, ms, kg, kN, GPa**

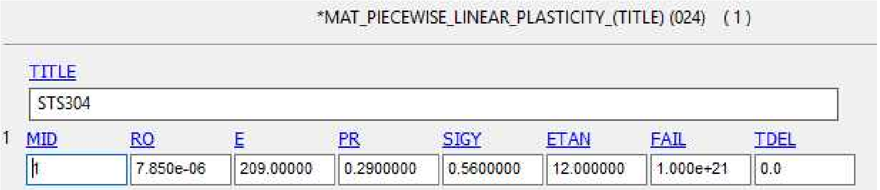


▲ Meshed Bogie Structure

Material Property

The Bogie structure is modeled as a shell element made of SAE4130 Steel. The physical properties model adopts **MAT24_Linear_Placticity**. And, the stress-cycle curve (**s-n curve**) required for fatigue analysis is set in *** MAT_Add_Fatigue**. There are various ways to set the s-n curve. There is a method of making a table directly and inputting it, also a method of inputting a graph through **Define_curve** and even a method of expressing a formula. In the **s-n** curve, **n** means cycle and **s** means stress level. In general, the s-n curve is expressed on a logarithmic scale. Structure Mesh Model

- **Keyword Manager → MAT | 024-Piecewise_Linear_Plasticity →Input properties**
- **Keyword Manager → Define Curve → create s-n curve**
- **Keyword Manager → MAT | Add_Fatigue → select curve in LCID**



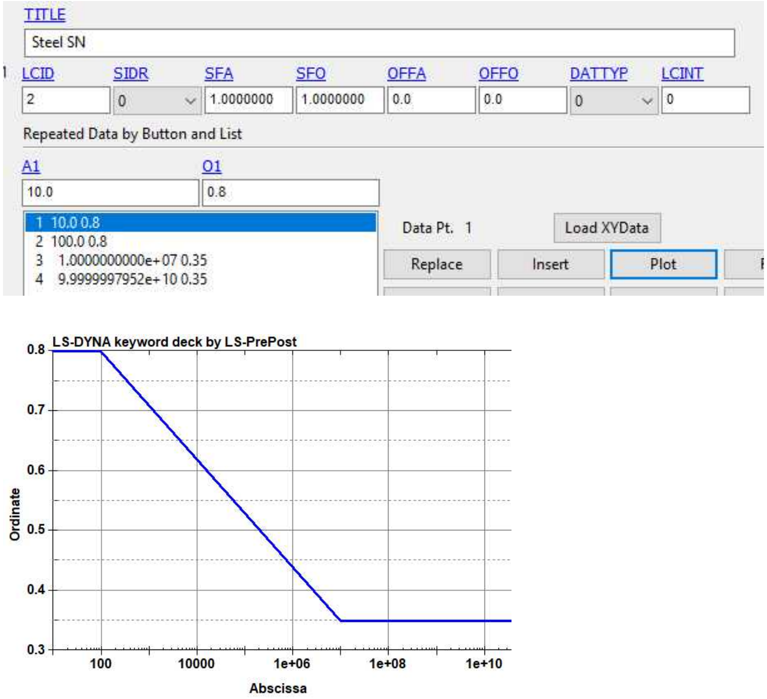
*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE

STS304

	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
1	1	7.850e-06	209.00000	0.2900000	0.5600000	12.000000	1.000e+21	0.0

▲ MAT_Linear_Plasticity Properties



TITLE

Steel SN

	LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	LCINT
1	2	0	1.0000000	1.0000000	0.0	0.0	0	0

Repeated Data by Button and List

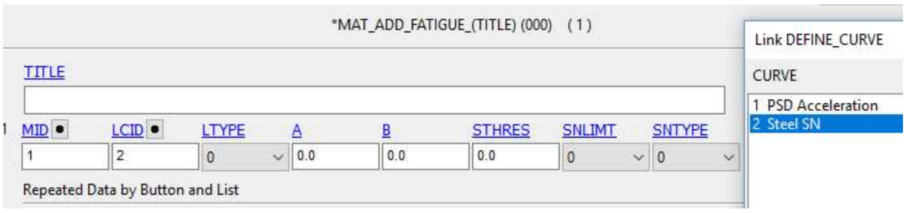
A1: 10.0

O1: 0.8

1 10.0 0.8
2 100.0 0.8
3 1.0000000000e+07 0.35
4 9.9999997952e+10 0.35

Data Pt. 1

▲ s-n curve data and graph for SAE4130 Steel



*MAT_ADD_FATIGUE_(TITLE) (000) (1)

TITLE

	MID ●	LCID ●	LTYPE	A	B	STHRES	SNLIMT	SNTYPE
1	1	2	0	0.0	0.0	0.0	0	0

Repeated Data by Button and List

Link DEFINE_CURVE

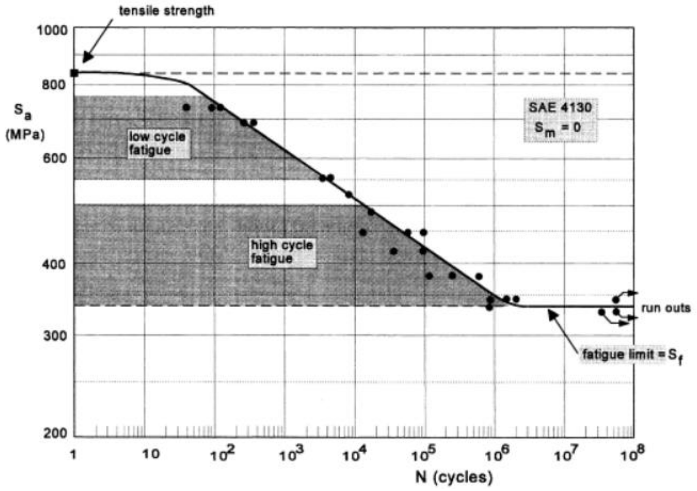
CURVE

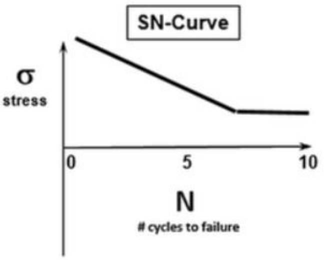
1 PSD Acceleration
2 Steel SN

▲ MAT_Add_Fatigue input

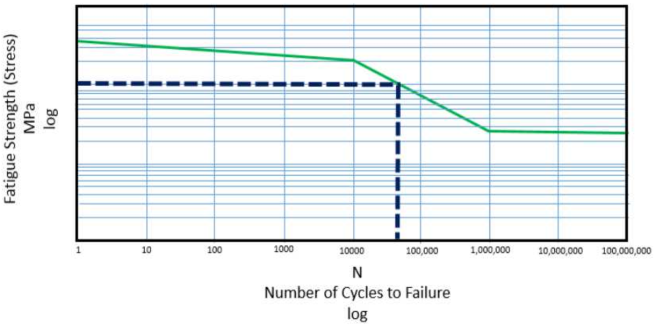
NOTE *s-n* curve

The figure below shows the s-n curve of the material SAE4130 steel used in this analysis. The ***s-n*** curve is the most important data for fatigue analysis. The x-axis is the cycle under load and the y-axis is the magnitude of the stress amplitude that acts repeatedly. To put it simply, it is a graph showing the correlation between the cycle and the magnitude of the stress amplitude until the structural member breaks.





The 's-n curve' was developed by the German scientist August Wöhler, while investigating a railway vehicle accident in 1842. At that time, the accident was found to be broken by repeated small loads acting on the Axle of the railway. The process of crack growth was proved by the s-n curve. This is almost the same situation as the one covered in this exercise. As shown in the figure below, it is a graph that can identify the period in which breakage occurs by knowing the magnitude of the amplitude of the applied load. This graph can be made through fatigue testing of many specimens, which differ slightly depending on the various international standards for the test. In LS-DYNA, the input is available by ***Define Curve** or **Equation**.

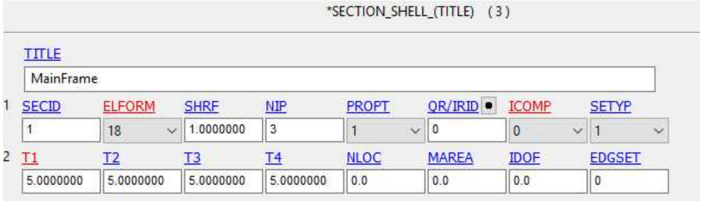


▲ Typical s-n curve

Element Property

After defining the properties, it should be assigned in the target part. First, section_shell is defined as follows for each part which is mainly modeled as shell elements. The thickness of each component is set to 5mm (Main frame), 4mm (cross and front beam) and 2mm (bracket). Once the material and section properties have been defined, set the material properties and section properties for the part.

- **Keyword Manager→Section_Shell→ELFORM=18, NIP=3, input 5, 4, 2mm repeatedly**
- **Keyword Manager → Part → SECID, MID**



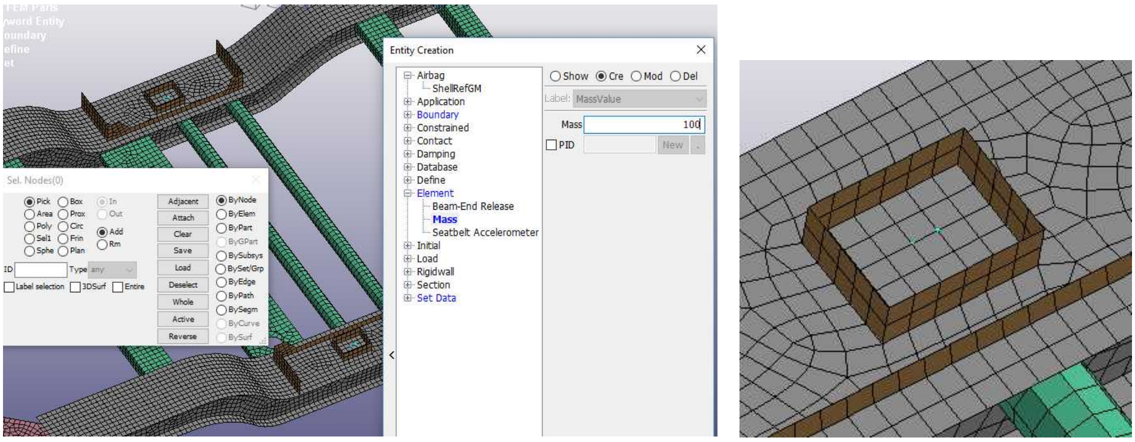
*SECTION_SHELL_(TITLE) (3)

TITLE
MainFrame

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
	1	18	1.0000000	3	1	0	0	1
2	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
	5.0000000	5.0000000	5.0000000	5.0000000	0.0	0.0	0.0	0

As shown in the figure below, select and fix each of the nine nodes of the four areas where the front and rear wheel-axles of the main frame are mounted.

- **Model and Part → Create Entity → Element | Mass → Sel. Node | Pick → Select center nodes 2 in Main Frame(see image) → Mass=100**



Entity Creation

○ Show ● Cre ○ Mod ○ Del

Label: MassValue

Mass 100

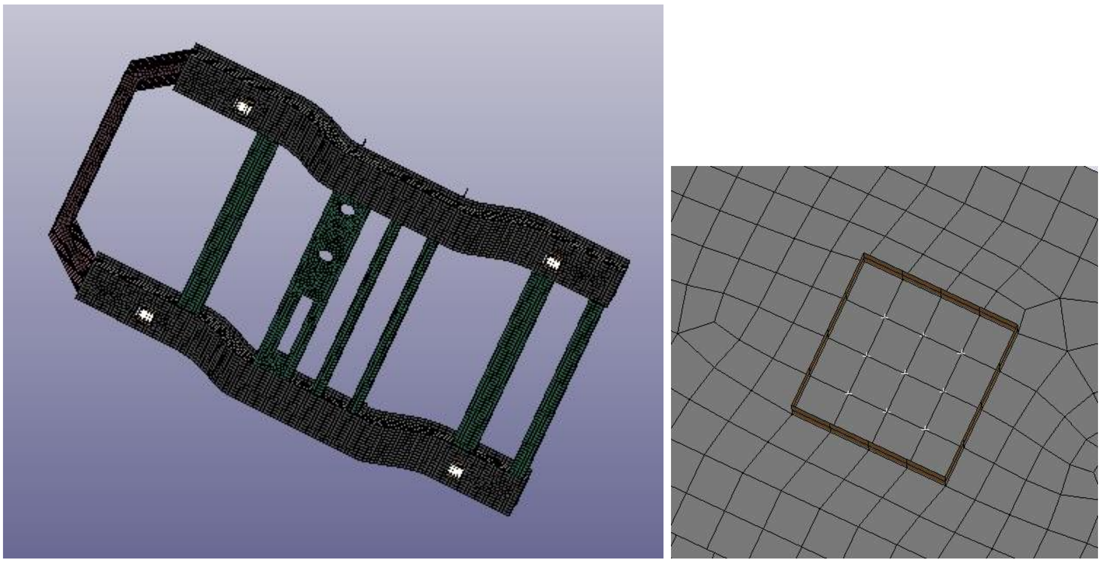
☐ PID

▲ Mass element setup

Boundary Condition

As shown in the figure below, select and fix each of the nine nodes of the four areas where the front and rear wheel-axles of the main frame are mounted.

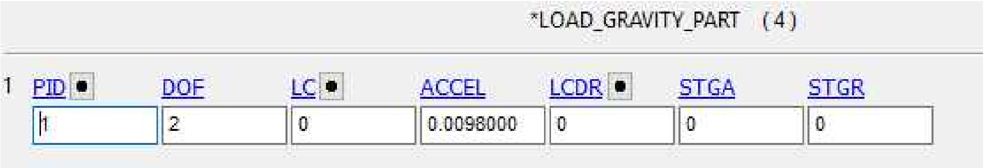
- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Pick → select nodes located in Frame(see image), 6 D.O.F all check → Apply**



▲ Boundary Condition Setup

Gravity acceleration is set to apply the effect of the self-weight of the bogie for the natural vibration analysis. Use the *** Load_Gravity** keyword to set the gravity acceleration acting on all four parts about the y-axis as shown below. Gravitational acceleration is 9.8m/s^2 = 0.0098 mm/ms^2, so please pay attention to the unit system.

- **Keyword Manager → Load → Gravity Part → DOF=2, ACCEL=0.0098 → Apply (Part_4)**



*LOAD_GRAVITY_PART (4)

	PID ●	DOF	LC ●	ACCEL	LCDR ●	STGA	STGR
1	1	2	0	0.0098000	0	0	0

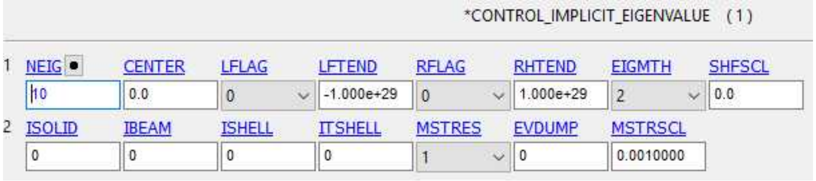
▲ Load_Gravity_Part Input

Resonant Frequency Analysis Setup

In the frequency-domain vibration durability analysis, the natural vibration mode must be analyzed beforehand. Natural vibration mode analysis is performed using the implicit approach. The natural vibration mode is analyzed in 10 modes of less than 1000Hz, which are the main vibration ranges of the railway and the extracted stress is applied in the endurance analysis.

- **Keyword Manager → Control_Implicit_Eigenvalue → NEIG =10, MSTRESS=1**

* NEIG → Calculate the number of analysis modes of natural frequency, initial 10, MSTRESS ► Select stress calculation (1 = calculation)

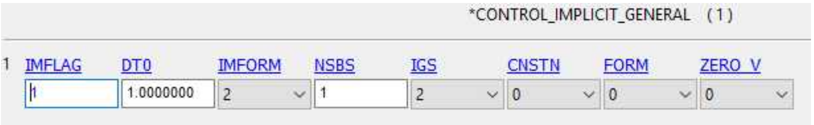


*CONTROL_IMPLICIT_EIGENVALUE (1)

1	NEIG ●	CENTER	LFLAG	LFTEND	RFLAG	RHTEND	EIGMTH	SHFSCL
	10	0.0	0	-1.000e+29	0	1.000e+29	2	0.0
2	ISOLID	IBEAM	ISHELL	ITSHELL	MSTRES	EVDUMP	MSTRSCL	
	0	0	0	0	1	0	0.0010000	

- **• Keyword Manager → Control_Implicit_General → IMFLAG=1, DT0=1, IMFORM=2**

  *** IMFLAG ► slect analysis type, Implicit=1 | DT0 ► Initial step size1 ms**

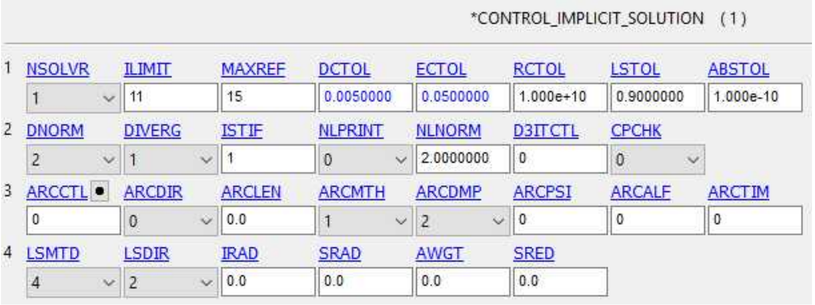


*CONTROL_IMPLICIT_GENERAL (1)

1	IMFLAG	DT0	IMFORM	NSBS	IGS	CNSTN	FORM	ZERO_V
	1	1.0000000	2	1	2	0	0	0

- **Keyword Manager → Control_Implicit_Solution → NSOLVR=1, DCTOL=0.005, ECTOL=0.05**

  *Type of Implicit Analysis (1 = Linear) | DCTOL, ECTOL ► Displacement and energy limits for convergence



*CONTROL_IMPLICIT_SOLUTION (1)

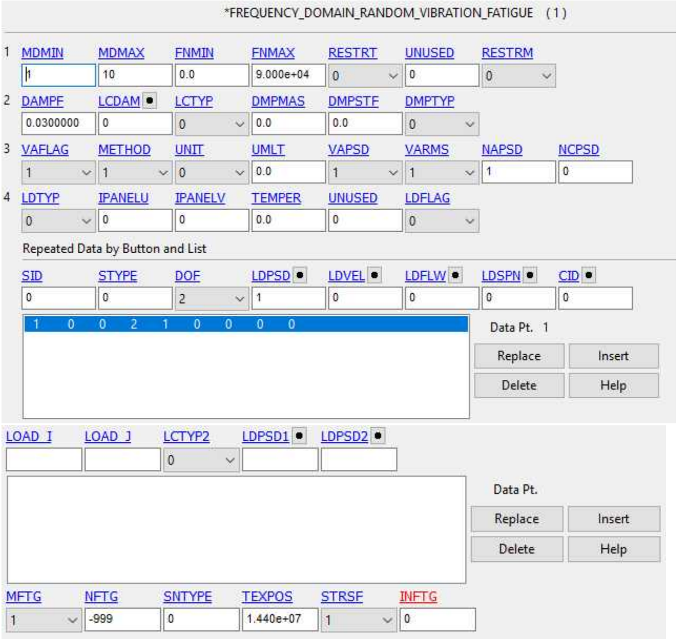
1	NSOLVR	ILIMIT	MAXREF	DCTOL	ECTOL	RCTOL	LSTOL	ABSTOL
	1	11	15	0.0050000	0.0500000	1.000e+10	0.9000000	1.000e-10
2	DNORM	DIVERG	ISTIF	NLPRINT	NLNORM	D3ITCTL	CPCHK	
	2	1	1	0	2.0000000	0	0	
3	ARCCTL ●	ARCDIR	ARCLEN	ARCMTH	ARCDMP	ARCPSI	ARCALF	ARCTIM
	0	0	0.0	1	2	0	0	0
4	LSMTD	LSDIR	IRAD	SRAD	AWGT	SRED		
	4	2	0.0	0.0	0.0	0.0		

▲ Natural Frequency Implicit analysis condition setting

Random Vibration Fatigue

This is the most important keyword in this exercise and it can be the most difficult part to understand if the reader does not have prior knowledge of fatigue analysis. As it is complicated, there are a lot of keywords cards involved. First, the **MDMIN** and **MDMAX** cards set the range of the vibration mode to apply to the endurance analysis. **FNMIN** and **FNMAX** likewise set the frequency domain to apply to the analysis. Since the unit of time is ms, pay attention to the input of the frequency unit. Since Hz, the unit of frequency, is 1 / s, 0.1 means 100Hz and 2 means 2000Hz when considering ms, which is the unit of this model. In the second line, the size of the damping is set. In this model, **DAMPF** (modal damping coefficient = 0.03) is applied.

- **Keyword Manager → Frequency | Domain_Random_Vibration_Fatigue → see image**



*FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE (1)

	MDMIN	MDMAX	FNMIN	FNMAX	RESTRT	UNUSED	RESTRM	
1	1	10	0.0	9.000e+04	0	0	0	

	DAMPF	LCDAM ●	LCTYP	DMPMAS	DMPSTF	DMPTYP		
2	0.0300000	0	0	0.0	0.0	0		

	VAFLAG	METHOD	UNIT	UMLT	VAPSD	VARMS	NAPSD	NCPSD
3	1	1	0	0.0	1	1	1	0

	LDTYP	IPANELU	IPANELV	TEMPER	UNUSED	LDFLAG		
4	0	0	0	0.0	0	0		

Repeated Data by Button and List

SID	STYPE	DOF	LDPSD ●	LDVEL ●	LDFLW ●	LDSPN ●	CID ●
0	0	2	1	0	0	0	0

1 0 0 2 1 0 0 0 0

Data Pt. 1

LOAD_I	LOAD_J	LCTYP2	LDPSD1 ●	LDPSD2 ●
		0		

Data Pt.

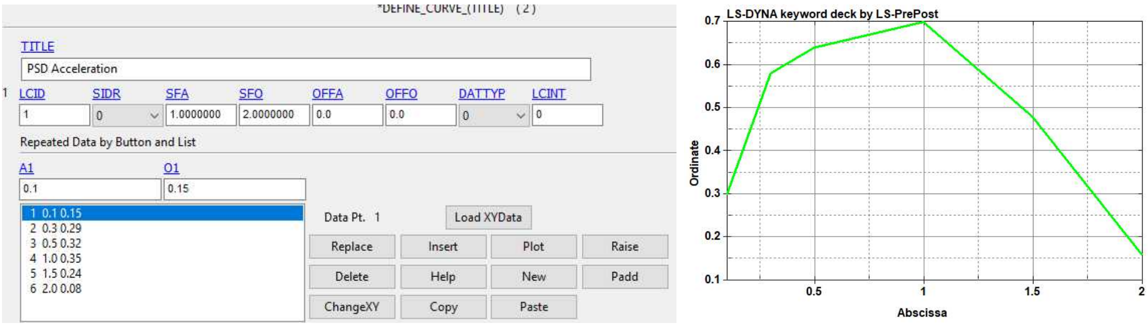
MFTG	NFTG	SNTYPE	TEXPOS	STRSF	INFTG
1	-999	0	1.440e+07	1	0

▲ Frequency_Domain_Random_Vibration_Fatigue Keyword

The third line sets the load application conditions. **VAFLAG** is the card about what kind of load to apply the vibration and **METHOD** is how to set up the mode response analysis. In this model, we will analyze using Acceleration (**VAFLAG** = 1) and **Superposition** Method (**METHOD** = 1). **VAPSD** and **VARMS** are to set the format for requesting PSD and RMS output. The fourth line is to set up the excitation area. **LDTYP** = 0 means excitation using the frequency domain PSD.

The table below shows the direction and size of the excitation. **DOF** = 2 means excitation in the y-axis direction and **LDPSD** is the card that sets the PSD for the excitation. Use **Define_Curve** to define the PSD you have. Create a PSD like the one below and configure it. The unit of **PSD** is g^2 / Hz and in this analysis, the unit of PSD unit is mm / ms^2. **SFO** is a safety factor for **PSD** size and 2 is applied in this model.

- **Keyword Manager → Define Curve → PSD, input following values**



*DEFINE_CURVE_(TITLE) (2)

TITLE: PSD Acceleration

	LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	LCINT
1	1	0	1.0000000	2.0000000	0.0	0.0	0	0

Repeated Data by Button and List

A1	O1
0.1	0.15

1 0.1 0.15
2 0.3 0.29
3 0.5 0.32
4 1.0 0.35
5 1.5 0.24
6 2.0 0.08

Data Pt. 1

▲ PSD curve generation

The last part is the area that sets the condition of the random vibration. **MFTG** is the part that sets the theory with random vibration, and **NFTG** is the part that defines how to set the ***s-n curve***. In this exercise, the **Steinberg 3 band** technique is applied, which is relatively simple to apply Random Vibration. Since the characteristic of the ***s-n curve*** of the material is already defined as **Add_Fatigue** in the material property, if -999 is set and the individual s-n curve is applied for each component, Table method should be applied. **TEXPOS** is the keyword to set the exposure time to random vibration. 1.44e + 7 ms means 4 hours of exposure.

NOTE **Steinberg 3 band theory**

The method of defining the random vibration load can be determined by various theories using the s-n curve rather than experimentally. In the LS-DYNA, various theories are provided. In this exercise, the Steinberg three-band method is applied.

- **Steinberg’s Three-Band Technique**
- **Dirlik method based on 4 moments of PSD**
- **Narrow Band Method**
- **Wirching Method**

The Steinberg three-band method is a very simple theory that expresses the stress cycle distribution as a Gaussian distribution and calculates the stress response amplitude levels in the 1 sigma, 2 sigma, and 3 sigma areas and combines them to extract von mises stress for fatigue analysis. For example, the method of extracting stress acting on fatigue analysis is as follows:

- **From 1 Sigma to -1 Sigma, operating at 68.3% of total time**
- **From 2 Sigma to -2 Sigma range, operating at 27.1% of the total time**
- **From 3 Sigma to -3 Sigma, operating at 4.33% of total time**
-

Therefore, the stress levels in these three bands are calculated as follows:

1 RMS X 68.3%
2 RMS X 27.1%
3 RMS X 4.3%

1 RMS stress is the von Mises stress obtained directly from the structural analysis and 2 sigma RMS and 3 sigma RMS are calculated using this.

The final cumulative fatigue damage index (***Rn***) is expressed as follows. As the theory is simple, there are many limitations, but it is a widely used technique for simply calculating fatigue life.

$$R_n = n\left(\frac{0.683}{N_1} + \frac{0.271}{N_2} + \frac{0.043}{N_3}\right) \textit{ Failure occurs when } R_n \geq C$$

Analysis Setup

The following step is to done to set the analysis condition. The analysis consists of two steps. In the first step, the natural frequency analysis is performed for 1 ms. In the second step, fatigue analysis is performed by superimposing the stress calculated in the random frequency analysis.

The fatigue life of the component calculated by the fatigue analysis is stored in a binary file called **D3FTG**. The following data is calculated and stored in the **D3FTG** file. Max Root Mean Square Stress, which is the stress level that calculates fatigue life, is calculated and stored in **D3RMS** file and **D3PSD** stores stress level by frequency in Power Spectrum Density calculated by analysis.

D3FTG: Cumulative damage ratio, Expected fatigue life, Zero-crossing frequency, Peak-crossing frequency, Irregularity factor, Expected fatigue cycles

- **Control | Termination → ENDTIM = 1, DTMIN=0.25**

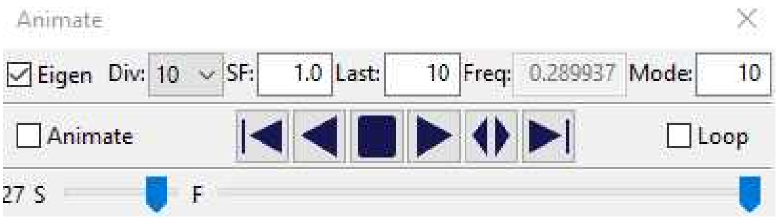
  * ENDTIM is the time to terminate the analysis. Implicit frequency analysis is 1ms.
- **Database→ D3plot → NPLTC=10**
- **Frequency_Binary_D3FTG → Binary =1**
- **Frequency_Binary_D3PSD → Binary =1, FMIN=0.1, FMAX=2, NFREQ=5**

  * Binary = 1 is the output calculation request, FMIN and FMAX are the frequency range of the output and NFREQ is the output cycle.
- **• Frequency_Binary_D3RMS → Binary =1**

This completes the process of generating a model for the natural frequency analysis and the random vibration fatigue analysis of the bogie structure of railroad cars. Save the generated model and compare to the provided model ***'W14_SETUP_Bogie_Random_Fatigue_R8.k'*** model.

Analysis Result

After the analysis is solved, open the **D3eigv** file created in binary format to see the results of the natural frequency analysis. Check **Eigen** in the **Animate** option and check the natural vibration mode with frequencies from 1st to 10th mode.

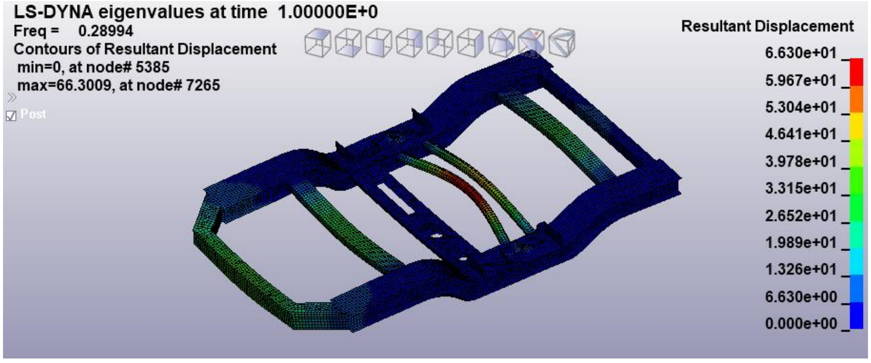


Animate

☑ Eigen Div: 10 SF: 1.0 Last: 10 Freq: 0.289937 Mode: 10

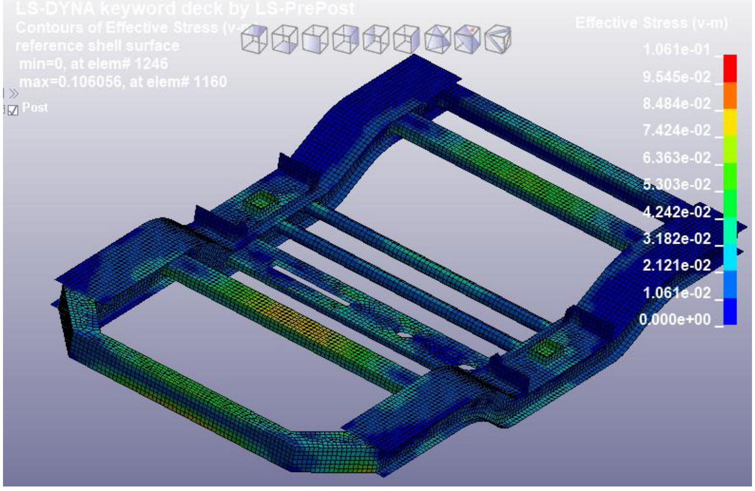
☐ Animate ☐ Loop

▲ Animate setup



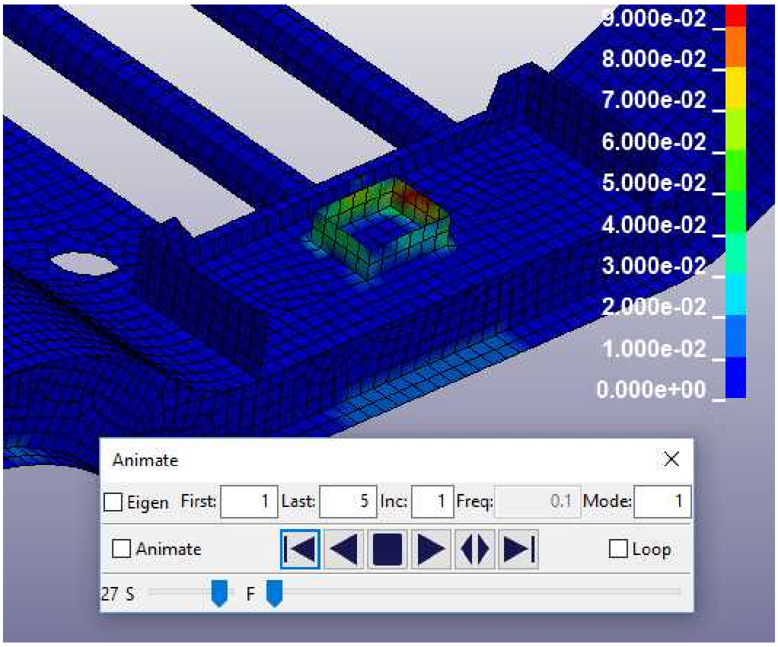
▲ 1st Mode shape at 289Hz

Next, open the **D3RMS** file to check the calculated RMS stress level. It can be seen that the maximum stress was calculated to be about 106 MPa occurring at the center of the cross beam. This stress value is used to calculate cumulative fatigue life through **Steinberg's theory**.



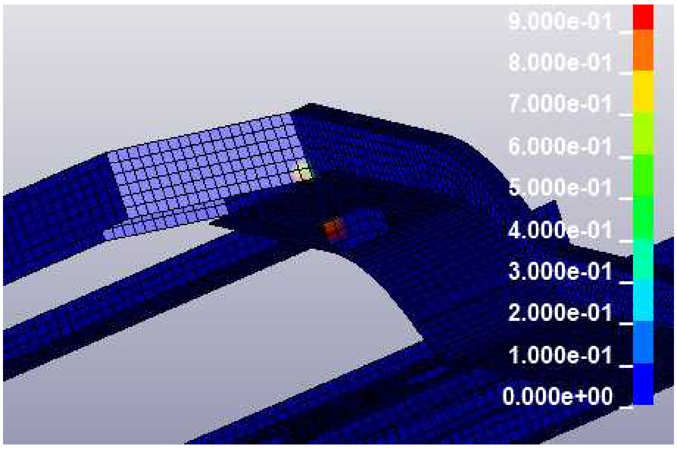
▲ RMS von Mises stress distribution

Next, let's check the PSD stress level. Since you entered **NFREQ** = 5 during the setup, the stress results in five frequency range between 100 Hz and 2000 Hz would have been calculated. By checking the individual stress levels, it can be seen that 166 MPa occurred in the bracket of the main frame at the frequency range where the highest stress occurred.



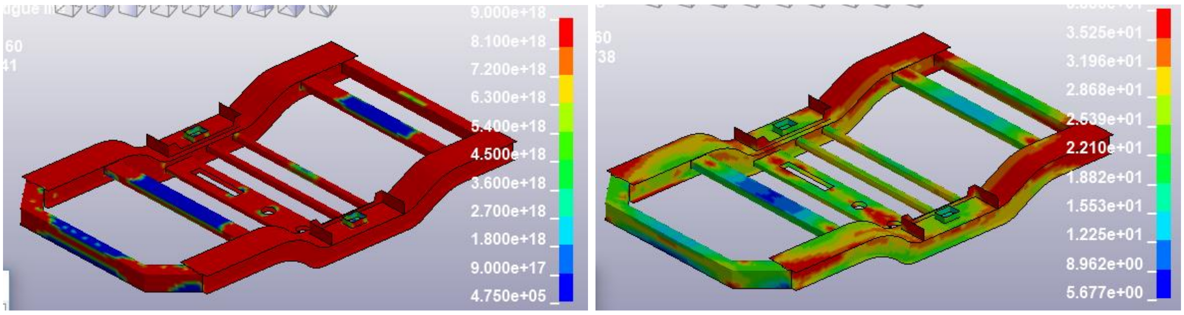
▲ PSD von Mises stress distribution at 100Hz

Open the most important **D3FTG** file in fatigue analysis. First of all, the cumulative damage ratio was calculated as the value of Damage 1 or more at the part where the front beam connected to the main frame and the bracket part. It is calculated by applying the Minor rule. If you select the **Safe / Failed zone** from the options, you can show the damaged elements in red and undamaged elements in blue based on Damage 1.



▲ Cumulative Damage Ratio Result

Next, check out the Expected fatigue life. As shown below, the calculated distribution from 4.7e + 5ms to 9e + 18ms (infinite life) is confirmed. Fatigue life itself has a very wide distribution in nature, so log-scale is mainly applied. It is easier to understand the fatigue life distribution by selecting a log10 scale from the **Fringe Component option** and plotting it on a log scale.

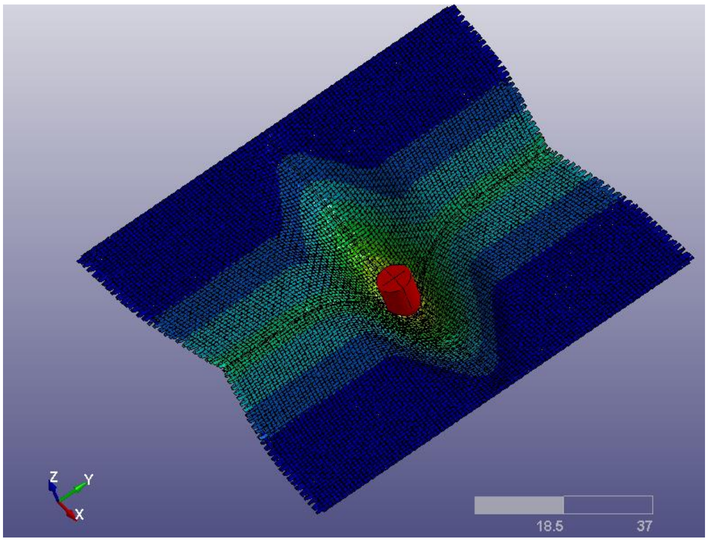


▲ Expected Fatigue life (log scale: Right) result

In summary, the lifespan of the front beam connection area and the central area of the cross beam is about 105 cycles, which is shorter than that of other areas.

Thus so far, the frequency domain random vibration simulation provided by LS-DYNA has been analyzed for the bogie structure of railroad vehicles and the fatigue life of each part is calculated to identify the parts requiring durability supplementation. In general, when designing a structure exposed to vibration, analysis, and verification of fatigue characteristics are often overlooked, while intensively analyzing dynamic behavior based on strength and strain analysis. The fatigue analysis field requires a lot of personal learning because it can be reliably analyzed only when sufficient background information and a clear understanding of the target structure's operating environment are supported.

Chapter 15. Ballistic Analysis of Woven Fabric



Objectives

- ✓ Learn the concept of bulletproof analysis
- ✓ Learn the structural modeling procedure of woven fabrics
- ✓ Understanding impact energy absorption mechanism.

Duration: 70min

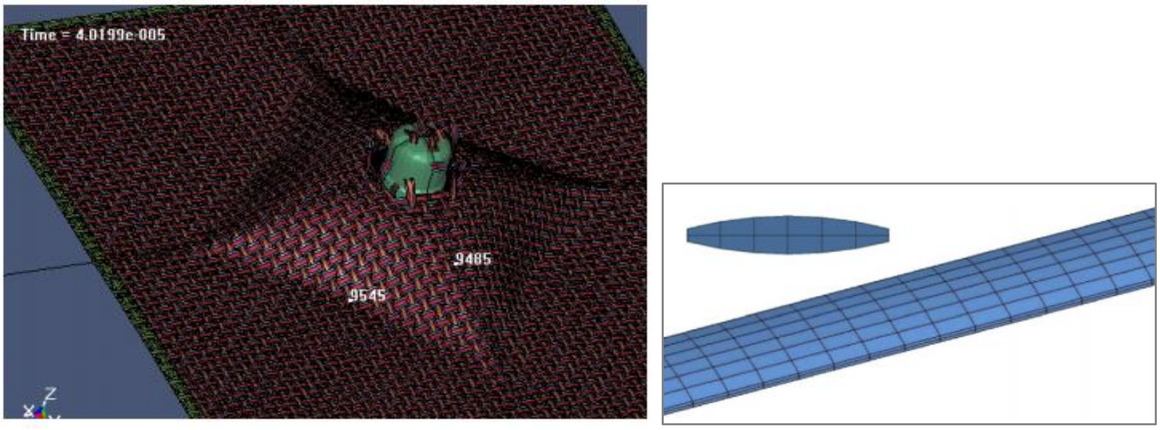
Application: Defense, Woven Fabric, Ballistic impact

Model Files

- W15_INITIALMODEL_Mesh.k: Initial state model
- W15_SETUP_Fabric_Impact_Model.k: Final state model

Introduction of Ballistic Impact

There's a high-strength fiber called Kevlar. It’s an Aramid-based fiber having very high strength and its flexible fiber makes it a better choice for personal protective clothing or shock absorbing structure in aviation or automobile. It is widely used in sporting goods, too. In the production of body armor using such fibers, analyzing the shock absorbing characteristics according to fiber woven pattern is a key field in the design of effective protective clothing. Various commercial products such as Kevlar, Xylon and Heracron are used in a wide range of applications. There are various analysis methods for high-speed impact analysis of such high-strength fibers. In this chapter, we will practice modeling techniques of ballistic impact analysis using woven aramid fibers.



▲ High-Speed Impact Simulation and Yarn Model for High-Strength Fiber Materials

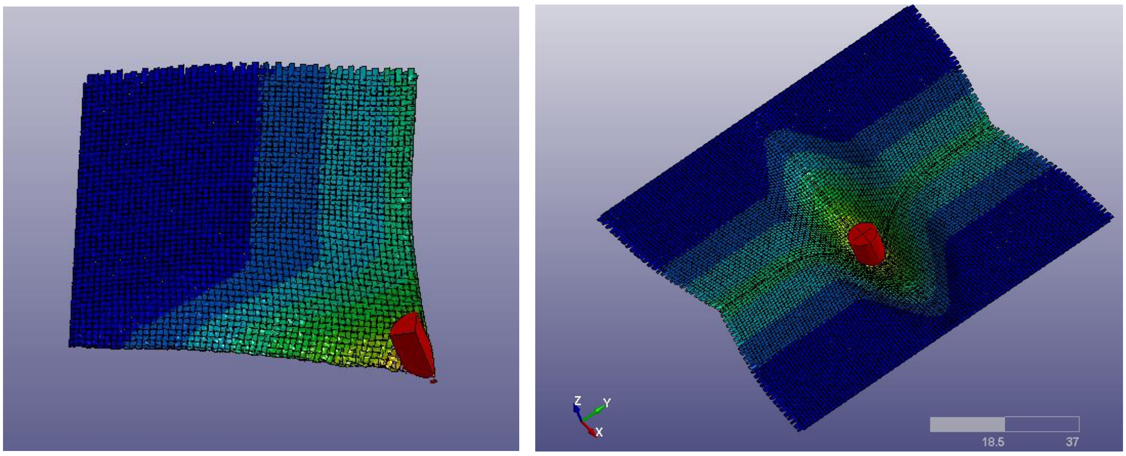
The friction between the fibers plays an important role during the shock absorbing mechanism of the woven fiber. Therefore, how to accurately simulate frictional force is a key part of modeling. In addition, modeling the weaving form of the fiber to match the actual fiber in the physical world is also an important factor in accurately simulating the shock absorption. However, the modeling of individual fibers results in a relatively long analysis time because numerous contacts occur between the fibers.

W15

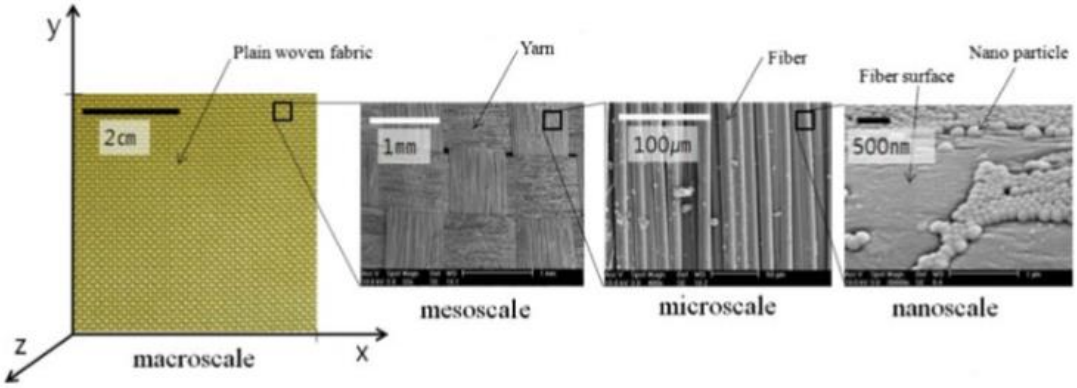
Workshop 15.
Ballistic Impact Analysis of Woven Fibers

Introduction

The analysis model consists of a Meso-Scale model in which individual bundles of yarns are woven in [0/90]s layup manner. In order to shorten the analysis time, the model was demonstrated using a symmetry modeling approach. The modeling of the fiber layer as a shell layer is called a macro-scale model. In this case, it is impossible to simulate the characteristics of the interaction occurring between the fibers. Recently, research on protective clothing has been actively conducted due to the increase in terrorist activity and a research paper for analyzing the bulletproof characteristics of such a high strength fiber can be easily found.



▲ Ballistic impact analysis training model (1/4 model and full model)



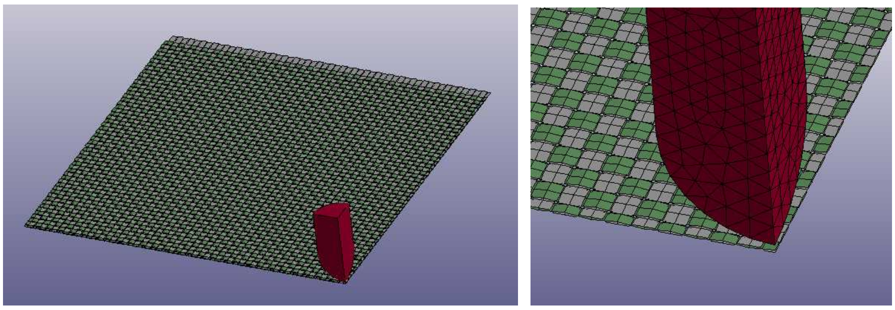
▲ High strength fiber modeling scale classification

In this workshop, we will practice 214m/s high-speed impact modeling for high strength fibers using the properties of Kevlar fiber, as well as the going thoroughly on overall analysis procedure, energy absorption mechanism and the influence of key parameters. The unit of analysis model is as follows:

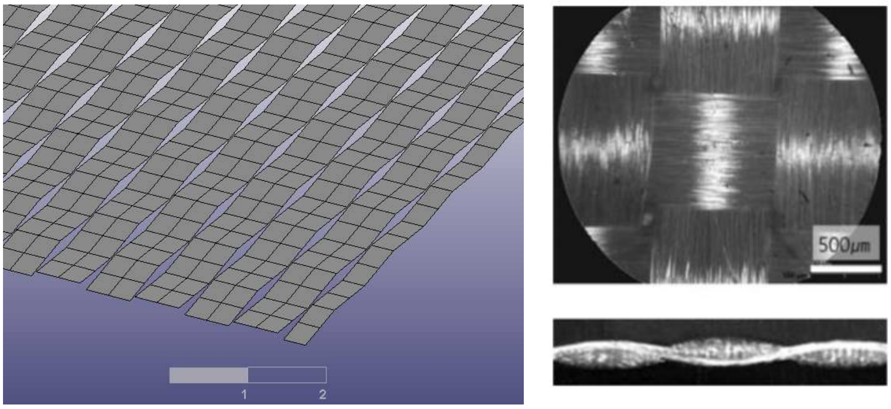
- **Unit: mm, ms, kg, kN, GPa**

Import Mesh

Import the model ***W15_Initialmodel_mesh.***k from which the mesh was created to analyze high-strength woven fiber high-impact analysis. The shape of the quarter model approach and full model which is of 50mm x 50mm fibrous layer, with the bullet will be displayed as shown below. Since each yarn is modeled as a shell element, if you look at the construction part, you will see that it consists of three parts: Yarn in the horizontal direction, Yarn in the vertical direction and a bullet which is the projectile impactor. Individual Yarn geometries were modeled using microscopically measured dimensions. One yarn is composed of two shell elements in the width direction. The smaller the shell element, the more accurate the contact or interaction can be simulated, but the appropriate level of the mesh should be created considering the analysis time.



▲ High-speed impact analysis mesh model (1/4 model)

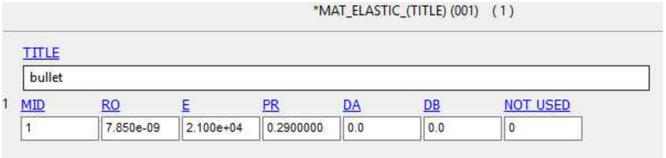


▲ Yarn part geometry and micrograph on the y-axis

Material Property

The bullet is assumed to be made of steel material and is composed of a solid model, with the property value input utilizing **MAT001_Elastic property.** Individual fiber layers modeled as Kevlar fibers are entered **using MAT003_ Plastic_Kinematic**. This property is obtained through actual experiments. The strength of the fiber is 3.9GPa and which has a Fail Strain of 0.045. Failure Strain value plays a role of zero stiffness by eroding an element when strain value reaches 0.045 due to high-speed impact. In general, *Etan* properties play an important role in defining plastic deformation in metals, but in the case of high strength fibers, plastic deformation does not occur and instead, fracture occurs. The actual tensile test results in linear deformation and ultimate fracture, so *Etan* should be set to zero in the physical properties. In fact, Yarn has the same anisotropic properties with different physical properties depending on the axial direction. However, in this high-speed impact event, the effect of the tensile stresses and deformation in that direction are dominant.

- **Keyword Manager → MAT | 001-Elastic →Input property for bullet**
- **Keyword Manager → MAT | 003-Plastic_Kinematic →Input property for fabric**

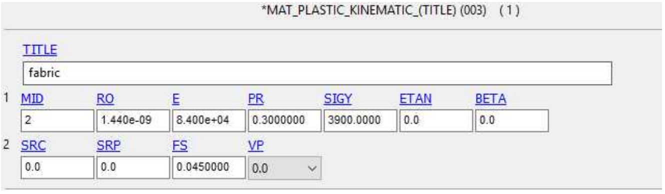


*MAT_ELASTIC_(TITLE) (001)   (1)

TITLE: bullet

	MID	RO	E	PR	DA	DB	NOT USED
1	1	7.850e-09	2.100e+04	0.2900000	0.0	0.0	0

▲ MAT001_Elastic Properties



*MAT_PLASTIC_KINEMATIC_(TITLE) (003)   (1)

TITLE: fabric

	MID	RO	E	PR	SIGY	ETAN	BETA
1	2	1.440e-09	8.400e+04	0.3000000	3900.0000	0.0	0.0

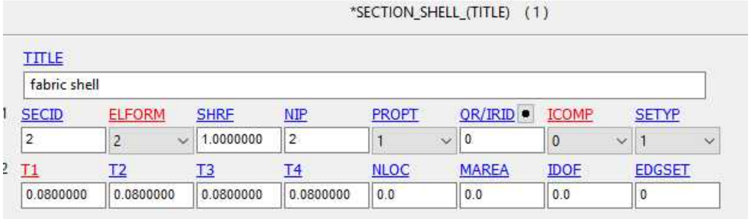
	SRC	SRP	FS	VP
2	0.0	0.0	0.0450000	0.0

▲ MAT003_Plastic Kinematics Properties

Element Property

After defining the Materials, the section properties should be defined in the target part. First, Section_shell is defined as follows on x-axis yarn and y-axis yarn composed of Shell elements.

- **• Keyword Manager → Section_Shell → ELFORM=2, NIP=2, Thickness=0.08**
- **• Keyword Manager → Section_Solid → ELFORM=1, AET=4**

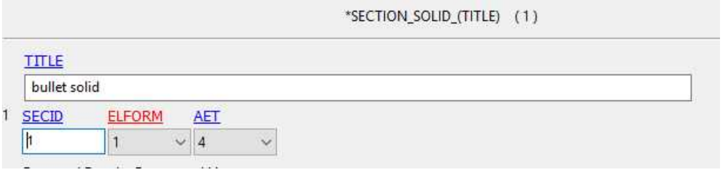


*SECTION_SHELL_(TITLE) (1)

TITLE: fabric shell

1	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID ●	ICOMP	SETYP
	2	2	1.0000000	2	1	0	0	1
2	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
	0.0800000	0.0800000	0.0800000	0.0800000	0.0	0.0	0.0	0

▲ Shell section properties for Yarn Layer



*SECTION_SOLID_(TITLE) (1)

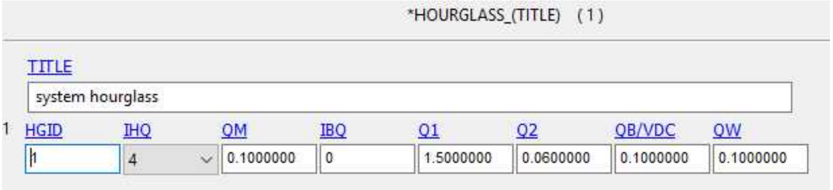
TITLE: bullet solid

1	SECID	ELFORM	AET
	1	1	4

▲ Solid section properties for Bullet

Before defining the characteristics of the part, create the Hourglass keyword. Hourglass is a phenomenon that causes abnormal deformation of the element mesh. Zigzag-shaped deformation appears and causes numerical errors during analysis. To prevent this, create the hourglass control **keyword** as follows: **Hourglass control** is described in detail later.

- **Keyword Manager → Hourglass → IHQ=4, QM=0.1 … input (see image)**



*HOURGLASS_(TITLE) (1)

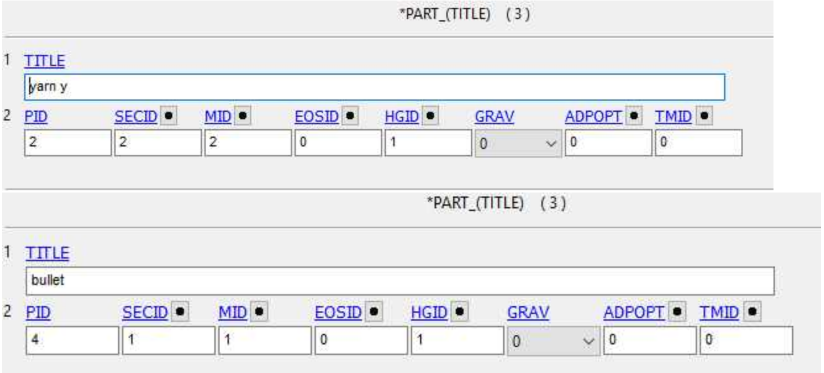
TITLE: system hourglass

1	HGID	IHQ	QM	IBQ	Q1	Q2	QB/VDC	QW
	1	4	0.1000000	0	1.5000000	0.0600000	0.1000000	0.1000000

▲ Hourglass energy Configuration

Once the properties, section properties, and hourglass have been entered, the properties and sections are defined for each part. The X-axis Yarn and the Y-axis Yarn are set identically.

- **Keyword Manager → Part →input SECID, MID, HGID (see image)**



*PART_(TITLE) (3)

1 TITLE: yarn y

2	PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
	2	2	2	0	1	0	0	0

*PART_(TITLE) (3)

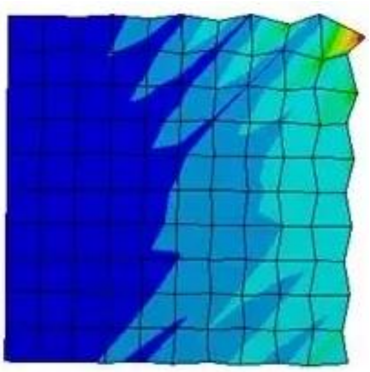
1 TITLE: bullet

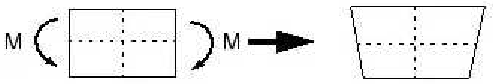
2	PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
	4	1	1	0	1	0	0	0

▲ Yarn and Bullet Part Properties Input Screen

NOTE About Hourglass Control

Hourglass is a technique named because the deformation of an element appears like an hourglass. Compared to the general macroscopic deformation, the deformation of individual elements is more severe in a zigzag form. It is mainly caused in the Hexagonal 3D element or 3D shell element. Check out the diagram below on the right. In a rectangular element with a net bending load, it's deformed at both edges, whereas the centerline is without deformation. If an element has a single integration point, zero energy mode phenomena appear, such as no stress occurring at the integration point. In other words, the strain energy is not generated. It is safe to say that the element in which this transformation occurs simulates a meaningless behavior.





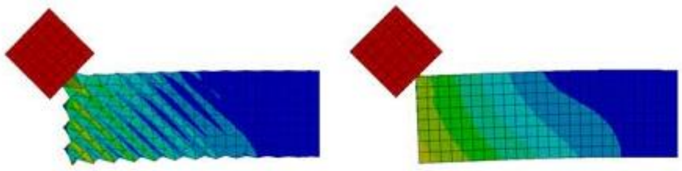
So how do you know if an Hourglass strain is occurring? The easiest way is to qualitatively check whether the deformation of individual elements is zigzag as shown above. Another is that hourglass energy for each part can be plotted by selecting matsum from the ASCII option in the analysis results.

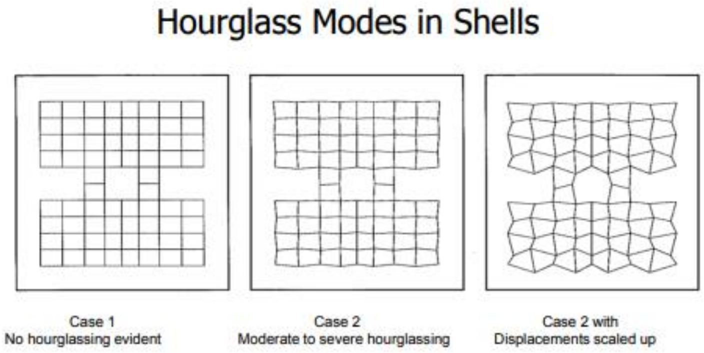
How can you control these hourglasses? The first to systematically prove the hourglass phenomena were Belytschiko and Flanagan in 1981. In this exercise, the Hourglass control keyword is used i.e. IHQ = 4, which prevents zigzag deformation by applying artificial stiffness to elements utilizing Belytschiko and Flanagan's theory.

So How can we control these hourglasses?

- The additional stiffness is set on the element to prevent hourglass deformation
- Inserting an artificial Viscosity on the element (typically the most widely used algorithm in high-speed impact models)

- Select the Fully integration element instead of the Reduced integration element.
- Reducing the size of the mesh may which is an easy solution.
- Use Viscos damping





Hourglass Modes in Shells

Case 1
No hourglassing evident

Case 2
Moderate to severe hourglassing

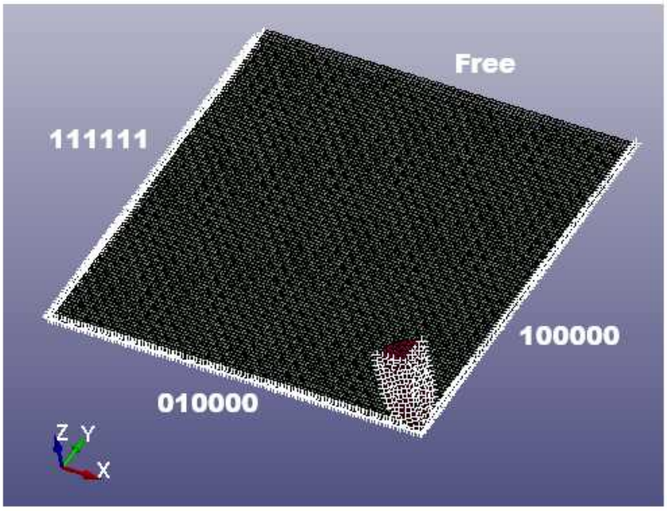
Case 2 with
Displacements scaled up

Hourglass energy shows the strain state of the element. Generally, it occurs at zero energy and is also called ZEM: Zero Energy Mode. It can be said that it occurs mainly at one integral point. Let's remove the Hourglass keyword from this created model, proceed with the analysis solving and then compare the results with the models that comprised Hourglass to check on how the elements behavior change.

Boundary Condition

In the impact analysis of high strength fibers, boundary conditions are a significant factor in the energy absorption mechanism. Observing the results of the analysis well along, you can understand why. In this exercise, you will fix the boundary conditions on both ends only (fix all DOF) and leaving the other end free. For the ¼ model, set the conditions (100000) and (010000) for nodes that are symmetric about the x and y planes.

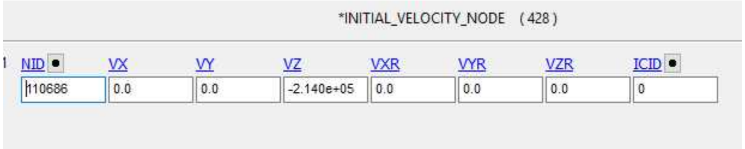
- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Area → Select each corresponding edge (see image) and setup DOF → Apply**



▲ Boundary Condition Setup

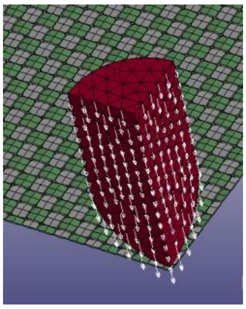
In Initial Velocity, define the bullet's impact velocity of 214m / s.

- **Model and Part → Create Entity → SPC | Cre → Sel. Node | Area → select elements of the bullet, input velocity(see image) → Apply**



*INITIAL_VELOCITY_NODE (428)

1	NID ●	VX	VY	VZ	VXR	VYR	VZR	ICID ●
	110686	0.0	0.0	-2.140e+05	0.0	0.0	0.0	0

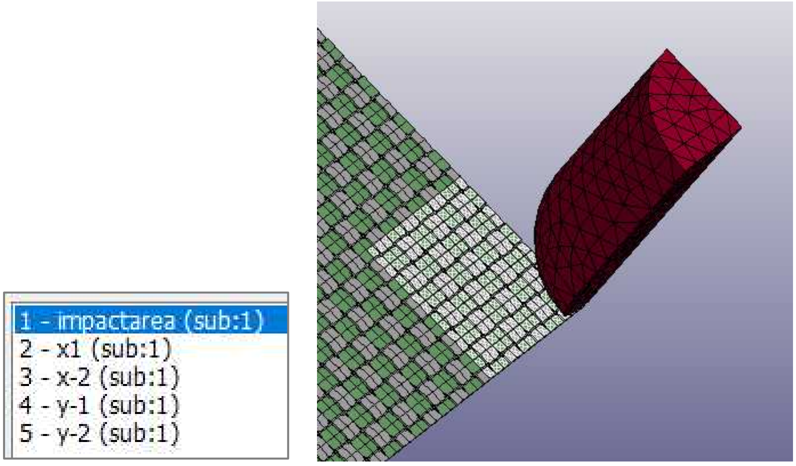


▲ Initial Velocity input

Contact Condition

Since the model includes individual yarns, proper contact condition setting is one of the important factors that determine the validity of the analysis. Because this model has to establish interaction conditions between Yarn and Yarn, '**Segment**' contact is used to establish effective contact conditions. In order to establish the contact between the bullet and the fiber layer which is as shown below, only the area where the contact occurs is first segmented with *impact_area.*

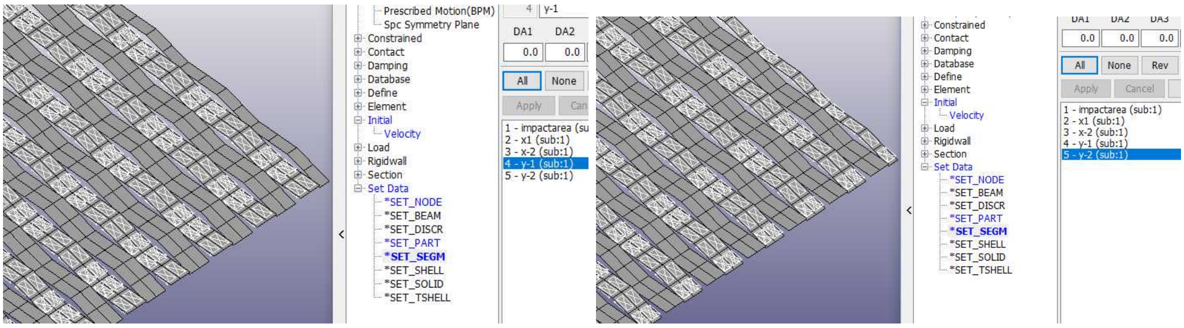
- **Model and Part → Create Entity → Set Data → Set_SEGM → Area | By element → select contact region elements (see image) → Apply**



1 - impactarea (sub:1)
2 - x1 (sub:1)
3 - x-2 (sub:1)
4 - y-1 (sub:1)
5 - y-2 (sub:1)

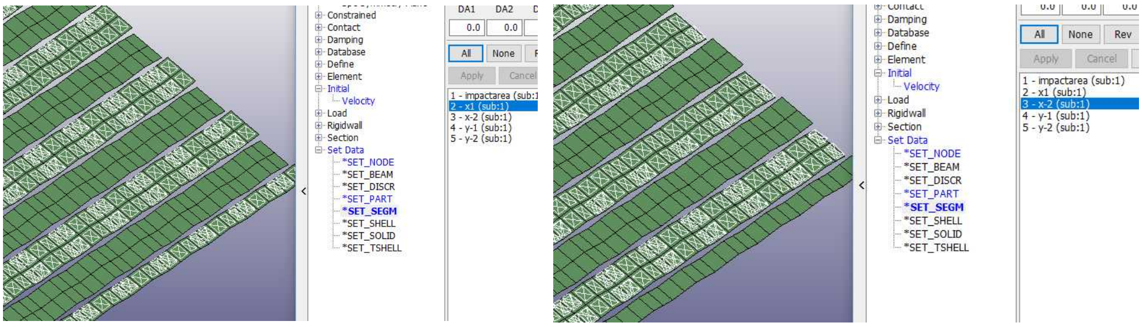
▲ Bullet contact area segment generation

Next, to establish contact between each individual Yarns and Yarns and also within the same Yarn part(similar to self-contact), select Yarns one by one as shown below and create segments with y1 and y2 respectively. Similarly, we create the yarn segments of x1 and x2.



1 - impactarea (sub:1)
2 - x1 (sub:1)
3 - x-2 (sub:1)
4 - y-1 (sub:1)
5 - y-2 (sub:1)

▲ Individual Yarn Segment generation (y1 and y2)

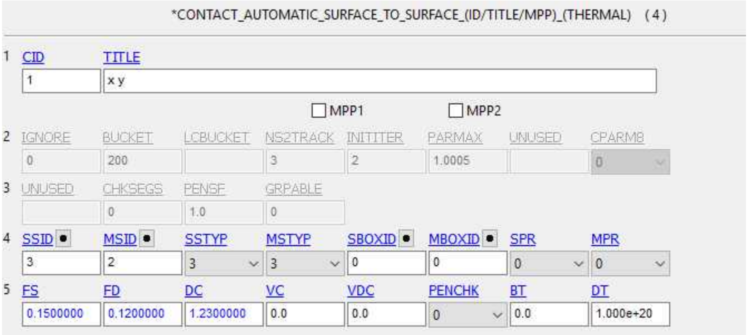


▲ Individual Yarn Segment generation (xl and x2)

Once individual Yarn segments have been created, the contact conditions can be set using ***Contact_Automatic_Surface _to_Surface**. The contact condition is set as follows.

- **Part (x-axis yarn) – Part (y-axis yarn)**
- **Part (Segment) – Segment (impact area)**
- **Segment (x1) – Segment (x2)**
- **Segment (y1) – Segment (y2)**

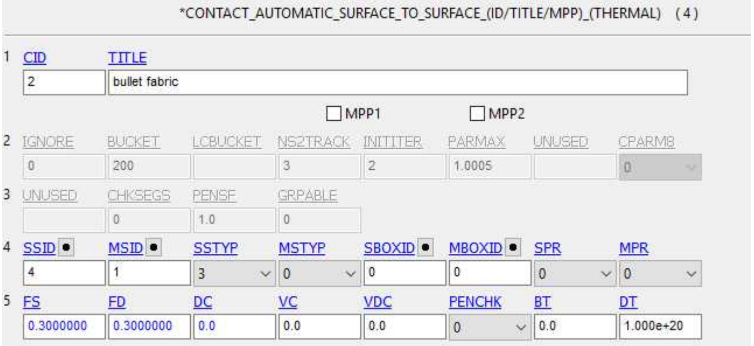
The dynamic and static friction coefficient should also be entered when setting the contact condition. Enter 0.15 for the static friction coefficient and 0.12 for the dynamic friction coefficient. These coefficients of friction are measured through yarn pull tests of actual Kevlar fibers.



*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (4)

1	CID	TITLE						
	1	x y						
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	3	2	3	3	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.1500000	0.1200000	1.2300000	0.0	0.0	0	0.0	1.000e+20

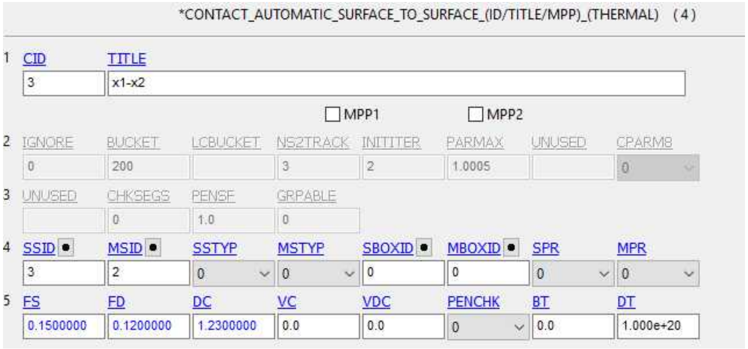
▲ Part (x axis yarn) – Part (y axis yarn)



*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (4)

1	CID	TITLE						
	2	bullet fabric						
				☐ MPP1		☐ MPP2		
2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	4	1	3	0	0	0	0	0
5	FS	FD	DC	VC	VDC	PENCHK	BT	DT
	0.3000000	0.3000000	0.0	0.0	0.0	0	0.0	1.000e+20

▲ Part (Bullet) – Segment (impact area)



*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (4)

CID	TITLE
3	x1-x2

☐ MPP1 ☐ MPP2

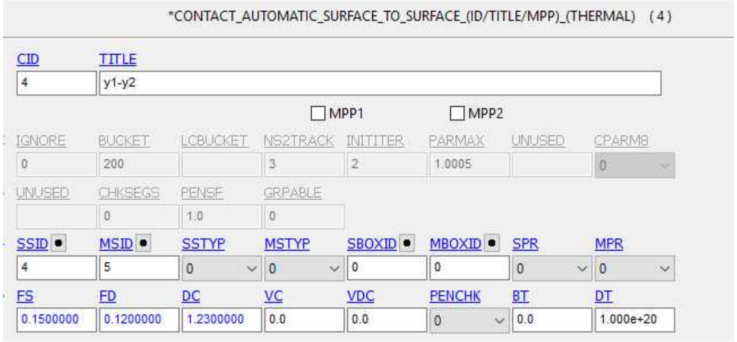
IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
0	200		3	2	1.0005		0

UNUSED	CHKSEGS	PENSF	GRPABLE
	0	1.0	0

SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
3	2	0	0	0	0	0	0

FS	FD	DC	VC	VDC	PENCHK	BT	DT
0.1500000	0.1200000	1.2300000	0.0	0.0	0	0.0	1.000e+20

▲ Segment (x1) – Segment (x2)



*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_(ID/TITLE/MPP)_(THERMAL) (4)

CID	TITLE
4	y1-y2

☐ MPP1 ☐ MPP2

IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
0	200		3	2	1.0005		0

UNUSED	CHKSEGS	PENSF	GRPABLE
	0	1.0	0

SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
4	5	0	0	0	0	0	0

FS	FD	DC	VC	VDC	PENCHK	BT	DT
0.1500000	0.1200000	1.2300000	0.0	0.0	0	0.0	1.000e+20

▲ Segment (y1) – Segment (y2)

Analysis Condition Setup

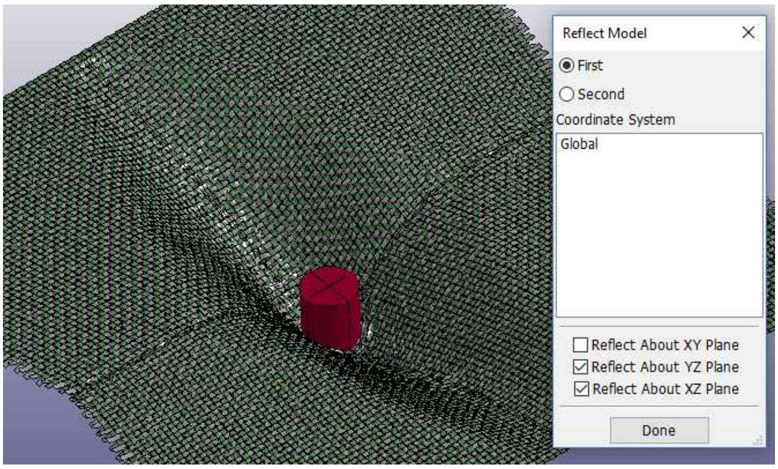
The following steps are done to set the analysis condition. The analysis end time is 8.5e-5 sec, which is until the bullet passes through the fiber.

- **Control | Termination → ENDTIM = 8.5e-5**
- **Database→ ASCII_option → DT=2e-7, GLSTAT, MATSUM, NCFORC**
- **Database→ BINARY_D3PLOT → DT=1e-6**

This completes the ballistic impact modeling of the bullet on the Kevlar fiber.

Result Visualization

The analysis will be completed within 30 minutes. Let's check the behavior of the bullet's high-speed impact and von mises stress at the region of the impact site. In **Model and Part**, select **Reflect** Model and click **Reflect** Model. The following dialog box appears. If you select YZ plane and XZ plane, you can display the 1/4 model as a whole model as shown in the figure.


Reflect Model

● First

○ Second

Coordinate System

Global

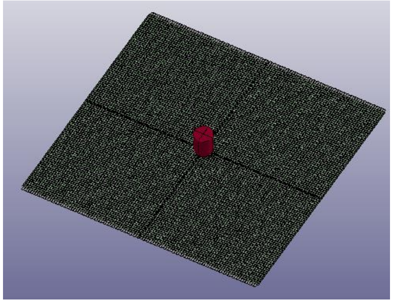
☐ Reflect About XY Plane

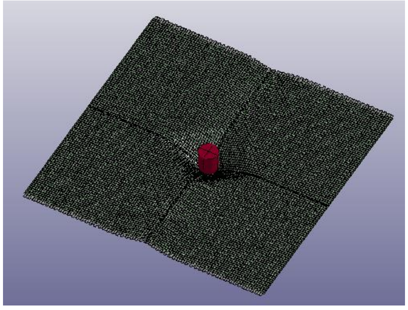
☑ Reflect About YZ Plane

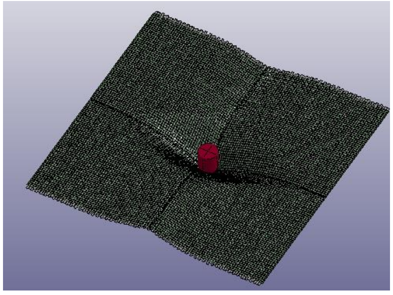
☑ Reflect About XZ Plane

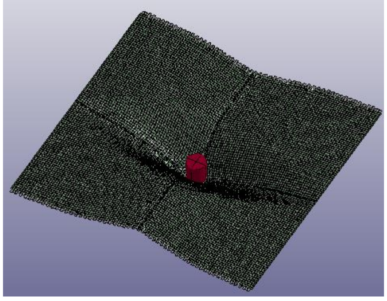
▲ Full Model visualization Using the Reflect Model Function

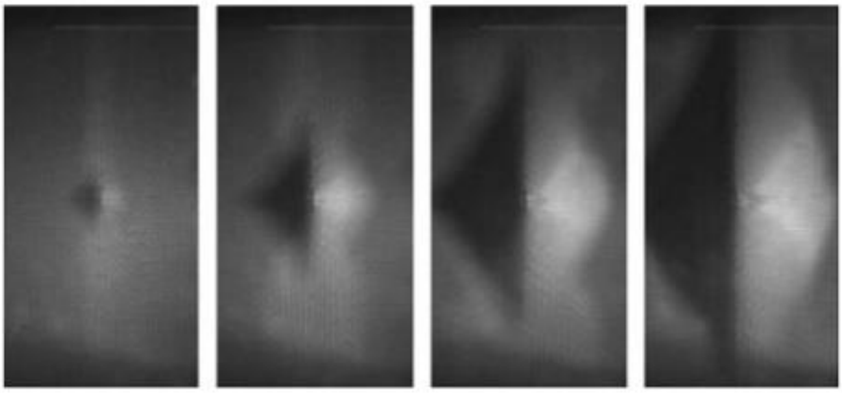
If you check the variation over time, you can see that the deformation occurs as the yarns on the free edge are pulled inward. The transformation of the kinetic energy of the bullet into the strain energy as the diamond-like deformation occurs overall. Looking at the rear face, it can be seen that through about 0.07sec, the penetration of the bullet begins as the yarns of the site where the direct contact with the bullet are broken/shattered. In fact, the high-speed impact test shows that the macroscopic deformation is almost similar to the deformation. It can be observed that in the deformation like diamond-type as well as this shape of the diamond along in the fixed end direction shows the same in both analysis and test results.


▲ 0.02sec

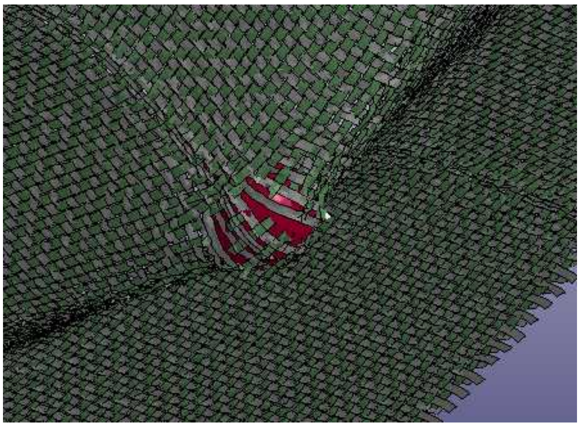

▲ 0.04 sec


▲ 0.06 sec


▲ 0.08 sec

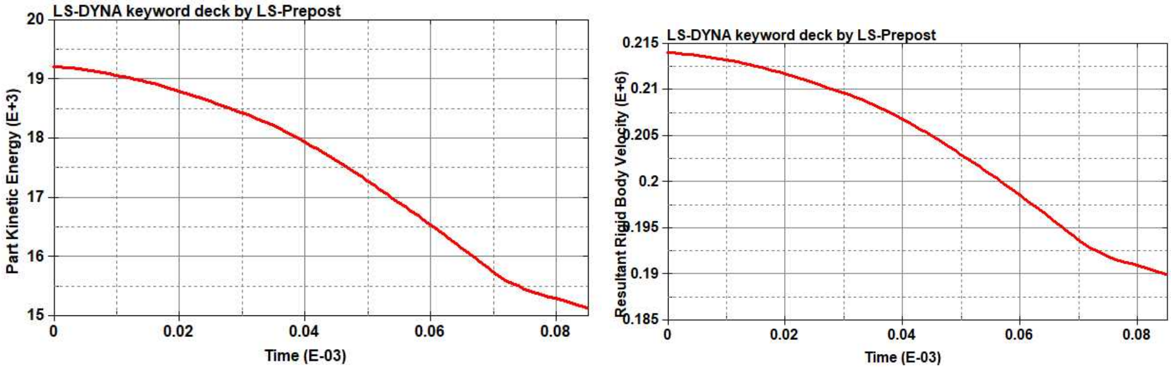


▲ Actual deformations taken using high speed camera



▲ Backface deformation and fibe breakage (0.08sec)

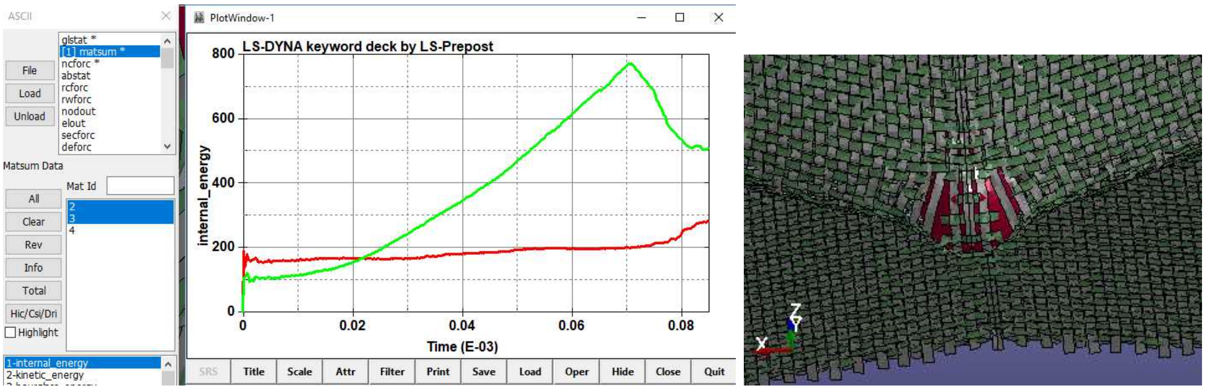
Let's check the change in the kinetic energy of the bullet and also the change in its speed. As expected, we can see that the velocity is reduced to 190 m / s, as the initial velocity of 214 m / s collides with the fiber, the kinetic energy is reduced to a similar trend. It can be seen that the point where the change in the kinetic energy and speed is abruptly relaxed around 0.07 seconds.



▲ Graph of kinetic energy and velocity of the bullet

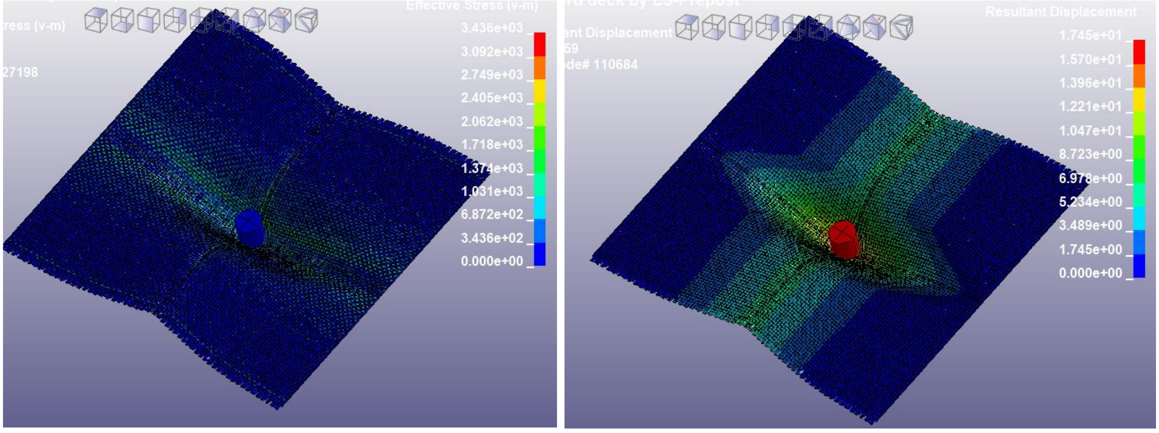
If you compare the internal strain energy of the x-axis Yarn and the y-axis Yarn, you can notice something interesting happening with boundary conditions. In the graph below, the green line highlights the x-axis yarn and the red line illustrates the internal energy of the y-axis yarn. The X-axis Yarn is a Yarn fixed at both ends, and the impact energy is rapidly increased as the bullet impacts, and the fracture energy is released when the fracture occurs at 0.07 sec. The increase is small and no breakage can be confirmed. This is where the boundary condition plays an important role in the impact energy absorption.

- **ASCII → matsum → Load, Mat ID=2, 3 | internal energy → Plot**



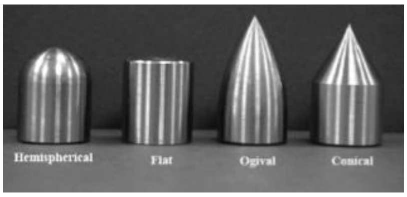
▲ Internal strain energy distribution of x-axis and y-axis yarn and backface deformation

While interpreting the analysis results, let's review the distribution of von mises stress and deformation. In fact, when compared with the actual impact test data taken with a high-speed camera, it can be seen that almost similar behaviors are simulated, such as overall deformation patterns and breakage points. Therefore, it can be confirmed that the modeling approach employed in this workshop is an appropriate model and it is actually applicable. Using the hand-on model, fix all four end edges, perform an analysis again and compare the results. It is easy to understand the difference between impact phenomena according to boundary conditions imposed.



▲ Von Mises distribution

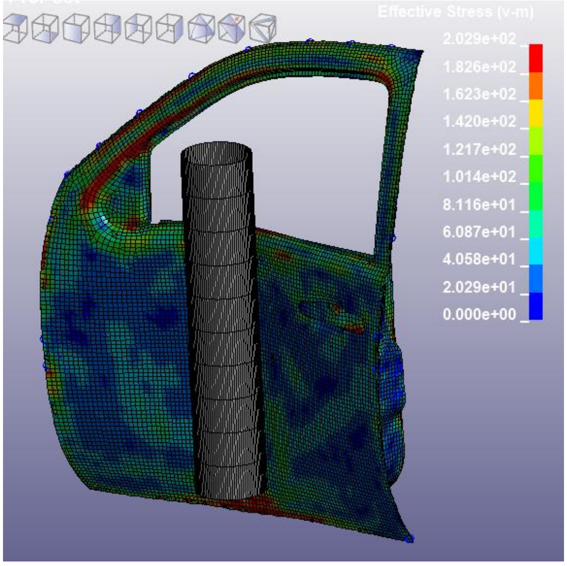
LS-DYNA has been used to model high-speed impact on high-strength woven fiber layers. This model employed individual yarns as shell elements, but there is no big difference from the results of the model comprised of 3D solid elements. On the other hand, considering that the 3D model takes almost 10 times more analysis time than the shell model, it can be seen that the Meso-Scale modeling strategy using the shell elements is very cost-effective.



▲ Various bullet head shape

Using a hands-on FE design, an evaluation of the energy absorption mechanism's complex behavior based on the various warhead shapes can help to understand the characteristics of the woven fabric.

Chapter 16. Door Impact Analysis with Spot Weld Joint



Workshop Objectives

- ✓ **Introduction of the theoretical background of Spot Welding**
- ✓ **Practicing Spot Weld modeling procedure**
- ✓ **Creating Door Impact model**

Workshop Duration: 80 min

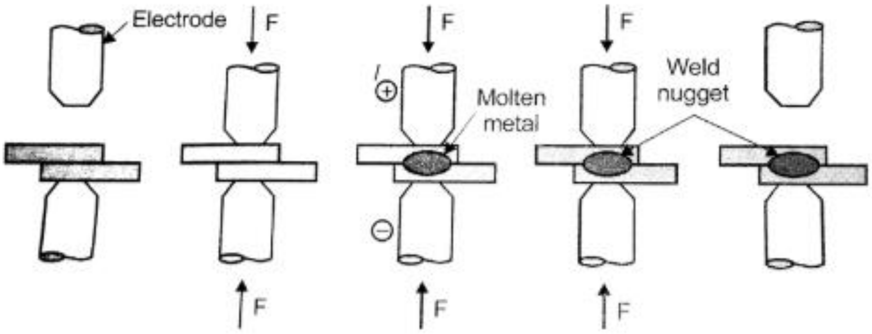
Application: Automotive, Spot Weld, Crashworthiness

Model Files

- W16_INITIAL_Spotweld_D1.k: Initial state model
- W16_Spotweld_D1.k: Rigid Spotweld model
- W16_Spotweld_E1/Fail.k: Deformable Spotweld model including failure model
- W16_SW_Door_Initial.k: Initial state of door impact model (mesh)
- W16_SW_Door_No Fail/Fail.k: The final state of door impact model

Introduction of Spot Weld

Spot welding is one of the oldest welding methods for joining metal plates using strong current and is widely used in all industries such as automobile, shipbuilding, and aerospace sectors. One of the welding types is electrical resistance weldings, generally used to weld 0.5mm to 3mm thick plates. As the name of welding says, Spot, it implies that a small current is applied to the high current of about 3kA ~ 100kA by instantaneously heating above the melting point (at that small zone) to join.



▲ Spot Weld Work Process

There are three main factors that affect the quality of spot welds: • Sheet metal thickness, • Electrode Force, Weld Current, and • Electrode Diameter. The most important thing out of these factors is the thickness of the target plate. If too thin, the welding does not occur or if it is too thick, the weld created is too weak and therefore strength it will be weak. As the metal is welded and recrystallized, the normal welding area occurs only when the diameter of the electrode is larger than the size of the nugget to be joined.

▲ Automatic Welding Machine

The spot welding process is particularly important in the automobile manufacturing process and many studies have been conducted on how to accurately simulate spot welds in the analysis process. This is a very important joining process, as SpotWeld has a dedicated program of analysis and design. Before the development of the analysis function, the welding part was modeled simply by tie connection or rigid connection. Recently, many functions to model deformation and breakage in the weld area have been introduced, and LS-DYNA also provides various elements and techniques for the Spot Weld model.

- **Rigid spotwelds**
- **Deformable spotwelds**
- **Spotweld failure**

Basically, the above three modeling techniques are utilized in general during analysis. The first is the rigid spotweld, which is simply connected. Secondly, there is a deformable spotweld method where deformation can occur, and a spot weld failure modeling method using several failure theories. In this exercise, we will study and practice this modeling technique for Spotweld applied to the vehicle door.

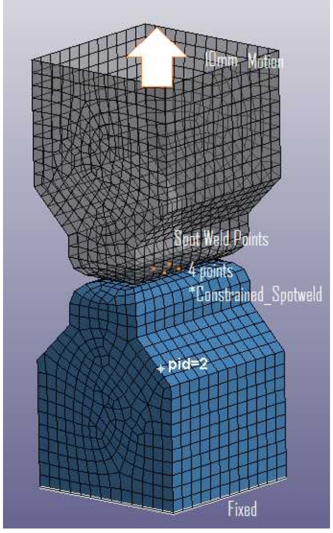
W16

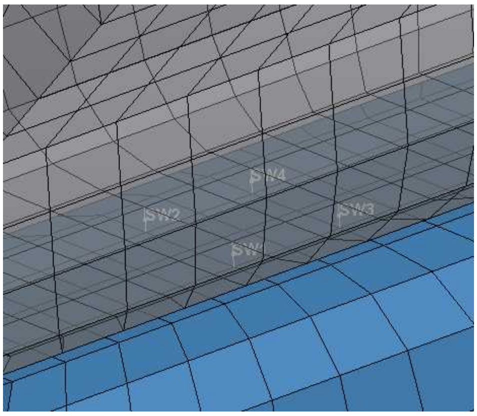
Workshop 16.
Impact Analysis of Spotwelded Door

Spot Weld modeling method – Rigid Spot Weld

Before we practice this model, let's take a look at the modeling technique of spot welds. It can be modeled with a rigid body and a deformable body as described previously. First, we will see how to model spot welds with a Rigid Model. We will practice using a model that joins the centers of two parts to each other by Spot Weld. There are two ways to model spot welds using rigid bodies. In this exercise, we will look at one of the ways to model Spot Weld which is the simplest and easiest. The disadvantage is that it can be welded only to the exact position of the node and various contact options or sophisticated damage models are not possible.

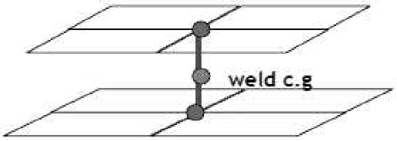
- **Method 1) *Constrained_Spotweld**
- **Method 2) *Beam_element and *MAT_rigid**





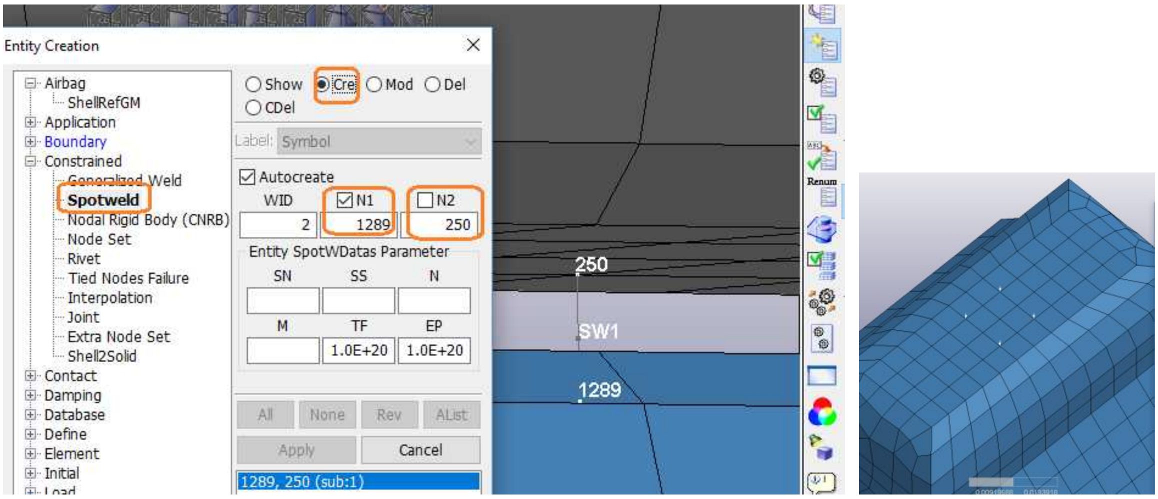
▲ *Constrained_Spotweld

First, let's open '***W16_Pre_spotweld_D1.k'*** which contains the basic modeling with mesh. Two jigs appear and basic elements with different boundary conditions (floor fixation) and load conditions (10 mm movement) are also modeled. Part of the exercise is concentrated only on the procedure for creating Spot Weld. You can define the point to be connected through Spot Weld by selecting the node to be connected as shown below. To input the failure criteria, you can enter the brittle failure criteria **SN** (normal force) and **SS** (shear force), or select **TF** (Failure Time) that defines the point of failure. In order to determine the damage, strain **EP** (plastic strain) can also be used as input.



- **• Create Entity → Constrained_Spotweld → Select the node to be connected as shown below**

  Enter the failure criteria (0 if not applied) → Done, create 4 Welds at the location



☑ Autocreate

WID	☑ N1	☐ N2
2	1289	250

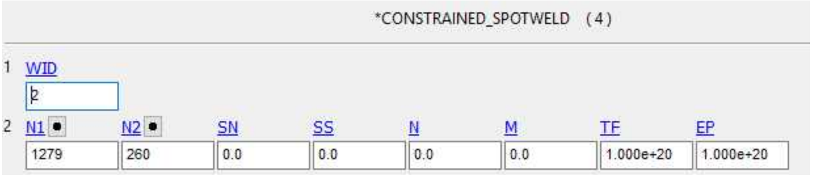
Entity SpotWDatas Parameter

SN	SS	N

M	TF	EP
	1.0E+20	1.0E+20

1289, 250 (sub:1)

▲ *Constrained_SpotWeld setup



*CONSTRAINED_SPOTWELD (4)

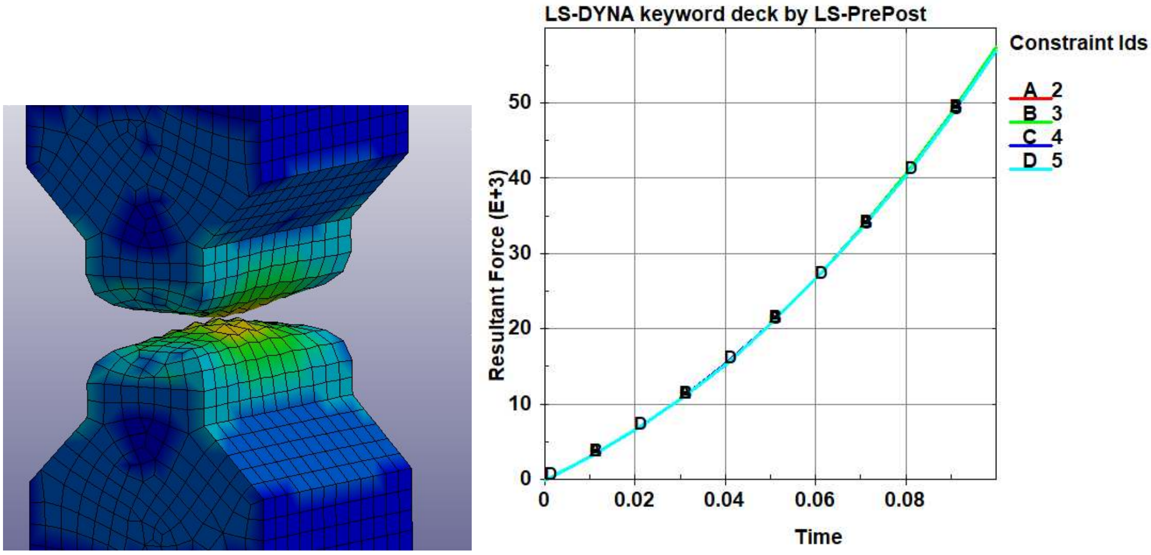
1 WID

WID
2

2

N1 ●	N2 ●	SN	SS	N	M	TF	EP
1279	260	0.0	0.0	0.0	0.0	1.000e+20	1.000e+20

After modeling spot welds, let's analyze and check the results. If you check the von mises stress, you can see that the upper jig moves while deformation occurs as follows. As the node of the spot welded part comes up, it shows a somewhat abnormal deformation. The application of spot welds of rigid body type requires the use of hourglass. Let's load **SWFORC** (Spot Weld Force) from the ASCII result file and plot the resultant forces of the four Weld objects. The linear loading results in an increase in working loads above 60 kN.



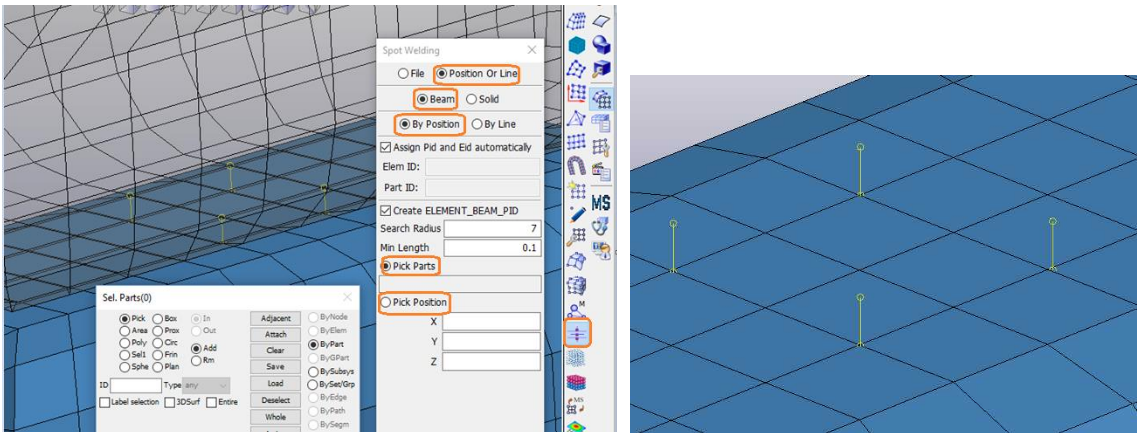
▲ Deformation of Spot weld area and SWFORC distribution

Spot Weld Modeling – Deformable Spot Weld

Now let's model the Deformable Spot Weld using the same model. Spot Weld of the elastic body where deformation occurs can be modeled using the Beam or Solid element. In general, the use of beam elements is recommended because the vehicle analysis involves a large number of spot welds modelization.

Let's recall the model we used earlier, '***W16_Pre_spotweld_D1.k***'. Create four Spot Welds using **Element tools → Spot Welding**. The welding beam thus created is generated as a single part. Since it is generated based on the position, it is irrelevant to the position of the node, so there is an advantage that the spot weld can be positioned at the desired position. However, if the mesh sizes of the parts to be joined are similar to each other to reduce errors in the joint analysis, create the same element size if possible.

- **Element and Mesh → Spot Weld → Position or Line | Beam | By Position → Check create element_beam_pid, Pick part (select parts 1 and 2) | Pick position Select 4 weld points in part 1**


Spot Welding

○ File ● Position Or Line

● Beam ○ Solid

● By Position ○ By Line

☑ Assign Pid and Eid automatically

Elem ID:

Part ID:

☑ Create ELEMENT_BEAM_PID

Search Radius: 7

Min Length: 0.1

● Pick Parts

○ Pick Position

X

Y

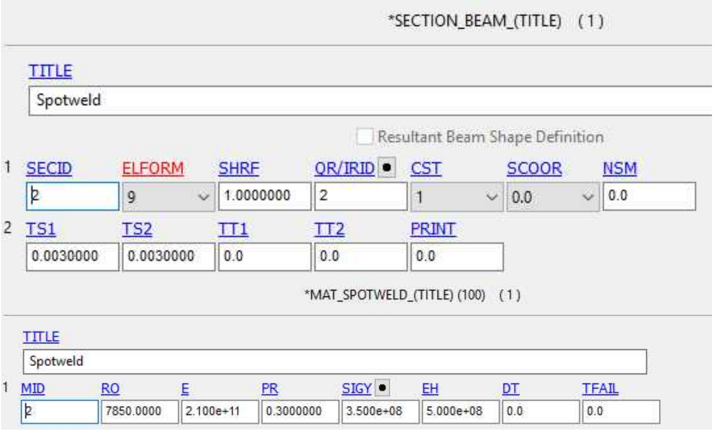
Z

▲ Spot Welding function

The next step is to define the properties of the beam element which is created as shown in the above step. The physical property is created by referring to the elastic with a plastic property of steel. **MAT100_Spotweld**, which defines the property of Spot Weld, is basically defined as Elastic-Plastic property. Through this beam element, the load/moment and stress are calculated at one Gaussian point and the validity of the **MAT100 Spot Weld** property model is shown in the technical data by comparing a number of experiments and analyses.

- **Keyword Manager → Section_Beam → Elform = 9, CST = 1 → TS1 = TS2 = 0.003**

  * The spot weld section must be ELFORM = 9, CST = 1 (Circular) which defines the cross-sectional shape of the weld nugget, the external diameter TS1 = TS2 = 3mm, and the internal diameter must be 0.
- **Keyword Manager → MAT_Spotweld → Properties of Steel Series (No Damage Criteria)**


*SECTION_BEAM_(TITLE) (1)

TITLE

Spotweld

☐ Resultant Beam Shape Definition

	SECID	ELFORM	SHRF	QR/IRID ●	CST	SCOOR	NSM
1	2	9	1.0000000	2	1	0.0	0.0

	TS1	TS2	TT1	TT2	PRINT
2	0.0030000	0.0030000	0.0	0.0	0.0

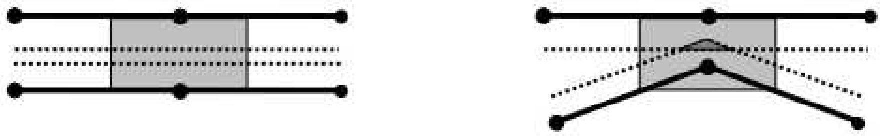
*MAT_SPOTWELD_(TITLE) (100) (1)

TITLE

Spotweld

	MID	RO	E	PR	SIGY ●	EH	DT	TFAIL
1	2	7850.0000	2.100e+11	0.3000000	3.500e+08	5.000e+08	0.0	0.0

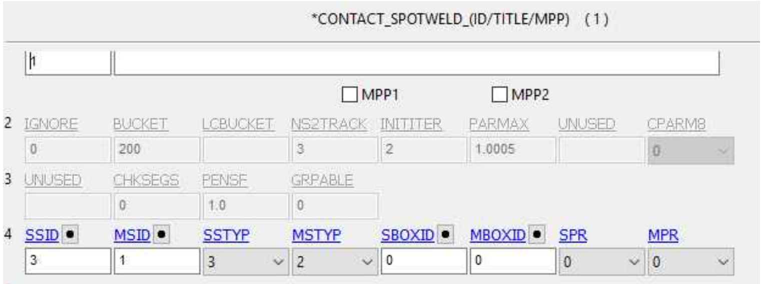
The next step is to enter the contact conditions. The relationship between the contact surfaces between the structures joined by welding is set. Generally, it is defined using the keyword ***Contact_Tied_Shell_edge_to_surface** or ***Contact_spotweld**. Select the target structure as the Master (**Part_set**) and the Spot Weld Beam part as a Slave (**Part**). In this definition, in the weld point, translational and rotation are interlocked in both parts and when the target structure invades the welding area by an external load, the contact force is calculated by penalty algorithm.



▲ Concept of contact force calculation in the welding area (Penalty Method)

- **Keyword Manager → Contact_Spotweld → Master = part set | Slave = spot weld → Done**

  *** Part set = Set top and bottom jigs to be welded as one set**



*CONTACT_SPOTWELD_(ID/TITLE/MPP) (1)

1

☐ MPP1 ☐ MPP2

2	IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
	0	200		3	2	1.0005		0
3	UNUSED	CHKSEGS	PENSF	GRPABLE				
		0	1.0	0				
4	SSID ●	MSID ●	SSTYP	MSTYP	SBOXID ●	MBOXID ●	SPR	MPR
	3	1	3	2	0	0	0	0

Let's save the generated model as '***W16_spotweld_E1.k'*** file and perform the analysis. It will take more time than the rigid body. Rigid body spot welds do not affect the time step of the analysis, but the spot weld modeled with elastic body affects the time step of the entire model, so mass scaling can be applied if necessary to reduce the analysis time. A global model can apply spot weld to **DT2MS** in ***Control_Timestep** and the time step can be input separately in DT parameter of ***MAT_spotweld** to apply Spot Weld element only. If Mass Scaling is applied to the entire model with **DT2MS**, separate spot weld adjustment is not recommended. Before examining the results of the analysis, let's create one more model with added failure criteria.

Spot Weld Modeling – Deformable Spot Weld (Failure)

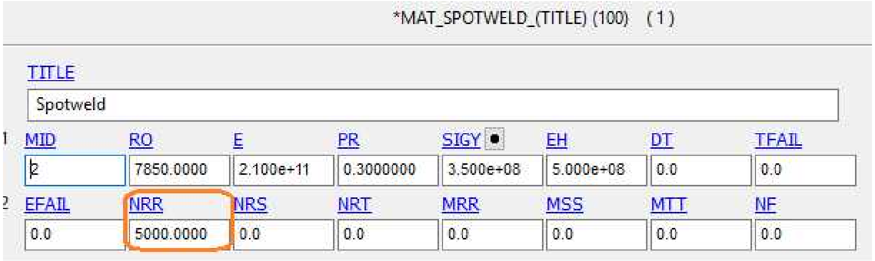
Add a failure criterion to calculate the spot weld failure and compare the results with the previous one. In many cases of welded mechanical parts, failure occurs at the weld zone. Although the latest welding technologies are commercially available, the welded area should not exceed 90% of the strength of the raw material. Of course, recrystallization in the heat-affected zone (HAZ) that occurs during the welding process, heat treatment, etc. may not occur properly, or there may be problems with metallurgical strength such as brittle fracture due to hydrogen inflow. The analysis only considers the mechanical part, so only the strength factor is considered. Failure theory that can be applied to Spot Weld can be applied to:

- **Time-based**
- **Plastic Strain**
- **Resultant Based**
- **Peak Axial Stress.**

Breakage strength is defined by the following formula. If only **Axial Force** (NRR) is input to 5,000N, define the spot weld to be removed when it works over this load.

$$\left(\frac{|N_{rr}|}{N_{rrF}}\right)^2+\left(\frac{|N_{rs}|}{N_{rsF}}\right)^2+\left(\frac{|N_{rt}|}{N_{rtF}}\right)^2+\left(\frac{|M_{rr}|}{M_{rrF}}\right)^2\left(\frac{|M_{ss}|}{M_{ssF}}\right)^2+\left(\frac{|T_{tt}|}{T_{ttF}}\right)^2-1=1$$

- **• Keyword Manager → MAT_Spotweld → NRR=5000 → Done**



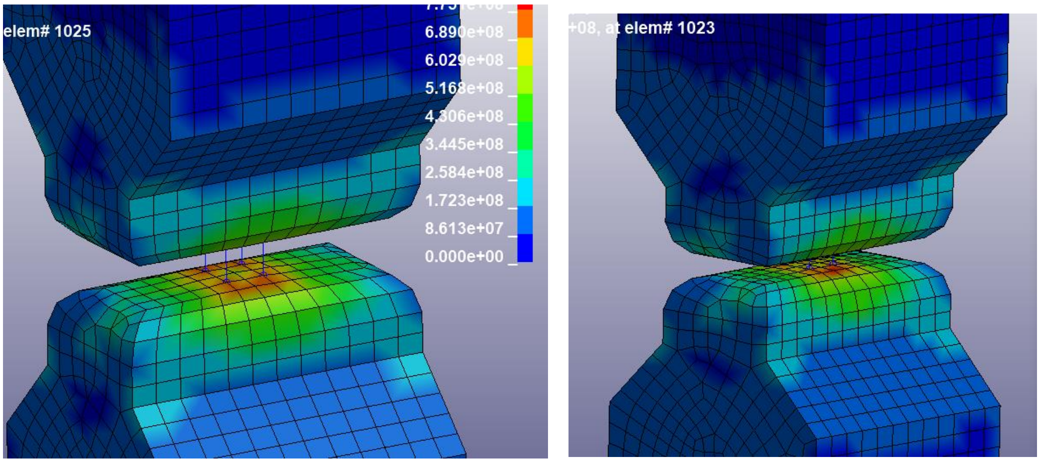
*MAT_SPOTWELD_(TITLE) (100) (1)

TITLE

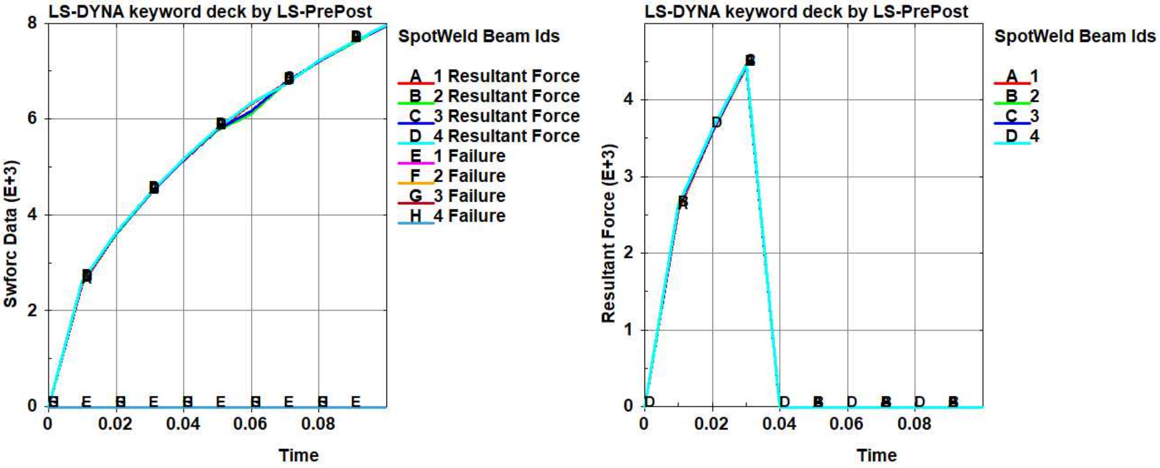
Spotweld

1	MID	RO	E	PR	SIGY ●	EH	DT	TFAIL
	2	7850.0000	2.100e+11	0.3000000	3.500e+08	5.000e+08	0.0	0.0
2	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
	0.0	5000.0000	0.0	0.0	0.0	0.0	0.0	0.0

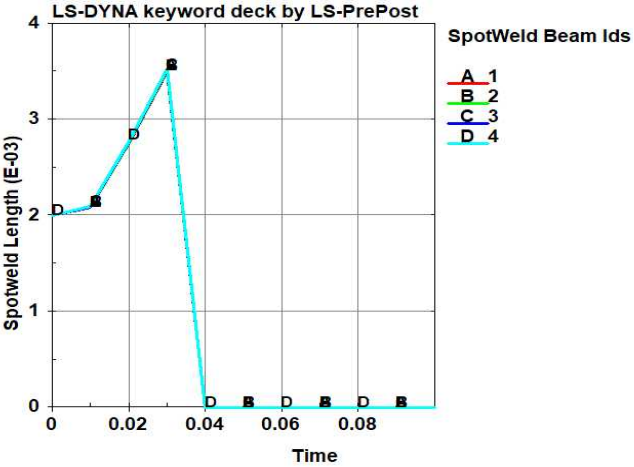
Let's save the model with the failure model added as ***'W16_spotweld_E1_Fail.k'*** and perform the analysis. Compare the results of the analysis with the models with and without damage. Models that do not contain damage are distorted as they move to the 10mm end. On the other hand, the model with damage reaches the Axial force of 5kN, which is the failure criterion after about 4mm movement and the weld beam element is deleted. Result file can be called by using ASCII variable from **DATABASE, SWFORC** (Spot weld force) can be requested to extract Axial forces, Spotweld length, Shear force, Resultant force and Failure from individual spot weld elements. The figure below compares the deformation, von mises and axial forces of the model without failure criteria and the added model. In the model which includes the failure criterion, the beam element is removed after about 20 frames and thus the bonding function is lost and the axial force is also reduced to zero.



▲ Weld Deformation and Von mises distribution (left) No failure (right) Failure (break after 20 frames)



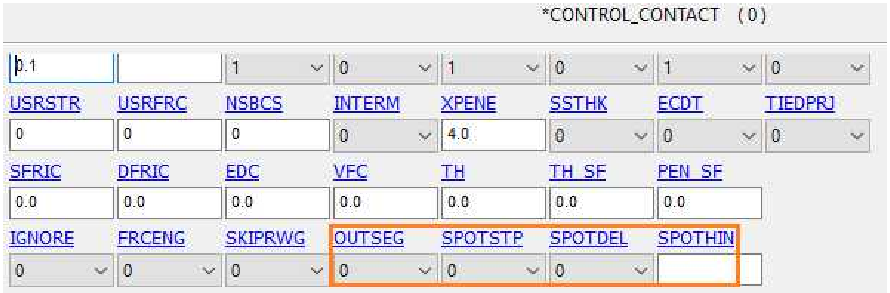
▲ Axial force distribution extracted from spot weld (left) No failure (right) Failure model



▲ Size of deformation of spot weld (Failure model)

So far we have discussed the basic spot weld modeling technique. Considering this technique, let's practice an example of collision analysis of a vehicle door. Other points to note during the Spot Weld modeling are as follows:

- Create an element size similar to the size of the target part being spot welded.
- The beam element size (TS1, TS2) of the spot weld should not be larger than the overall thickness of the panel to be welded.
- Place at least three or four elements on the face of the spot weld.
- Panels to be welded should be as parallel as possible. Spot Weld may not be applied properly between overly inclined panels.
- Define the contact part as spot weld separately from the contact setting part of the whole model.
- Note the ***Contact_Spotweld** setting. Spot Weld (Beam or Solid) should be Carefully set to one Part_Set for the welded per part. If you set parts that are not connected to each other, an error occurs and the analysis is interrupted. If you apply Spot Weld and if it is not connected to each other properly, again the error will occur in the analysis. This is caused by ***Contact_Spotweld** not being set properly.
- Parameters related to Spot Weld in ***Contact_Control** are as follows. The **OUTSEG** card is an option to extract information about all spot weld nodes and their interconnected elements. **SPOTTSP** is an option that determines whether the analysis should proceeds when the welded master element is damaged. **SPOTDEL** is an option that effectively reduces the computation time by deleting the associated spot weld element if the master element is broken off. **SPOTHIN** is a scale factor for the contact load calculation between the weld area and the target part, which means that the contact load is reduced by about 50% when 0.5 (50%) is input.



*CONTROL_CONTACT (0)

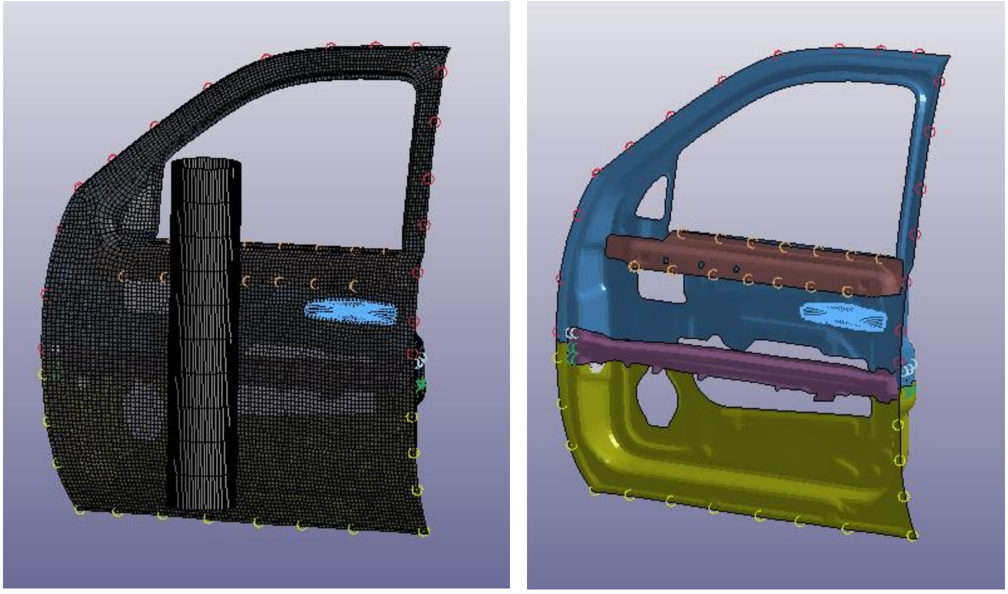
0.1		1	0	1	0	1	0
USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
0	0	0	0	4.0	0	0	0
SFRIC	DFRIC	EDC	VFC	TH	TH_SF	PEN_SF	
0.0	0.0	0.0	0.0	0.0	0.0	0.0	
IGNORE	FRCENG	SKIPRWG	OUTSEG	SPOTSTP	SPOTDEL	SPOTHIN	
0	0	0	0	0	0		

▲ Options in the * Control_Contact keyword related to spot welds

Import Door Impact Model

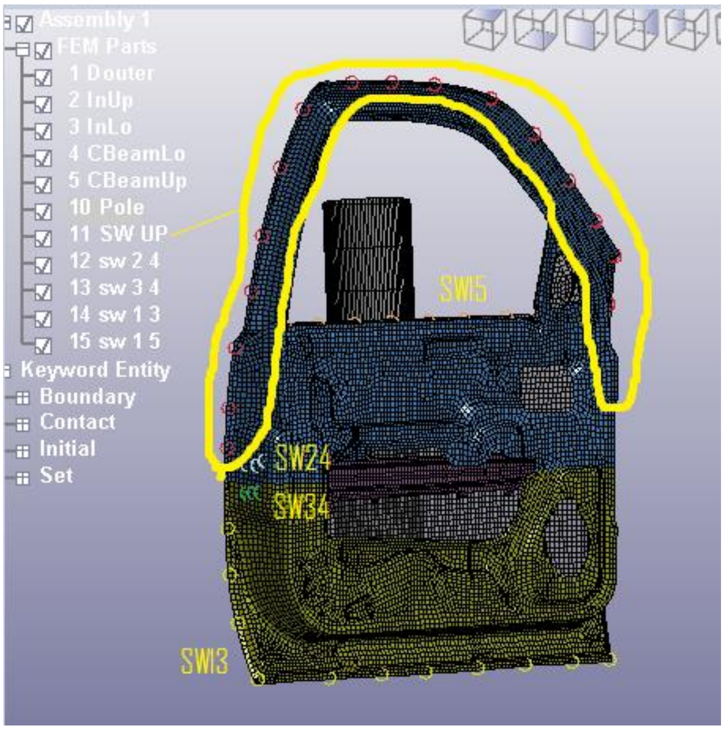
Import the ***'W16_SW_Door_Pre.k'*** model, which is the default model for the Door Impact analysis model. In this exercise, we will only practice on Spot Weld, but not on the whole modeling. The imported model has basic keywords and does not include other keywords except the partial spot welds. The figure below shows a model consisting of five imported Spot Welding Beam parts. The inside of the door is a framework where an impact beam is connected to the door panel by spot welding to withstand shock. The impact velocity is about 6m / s on the pole which is made up of a rigid body. The unit system of the model is as follows:

- **Unit System: mm, Tonne, sec, N, MPa.**



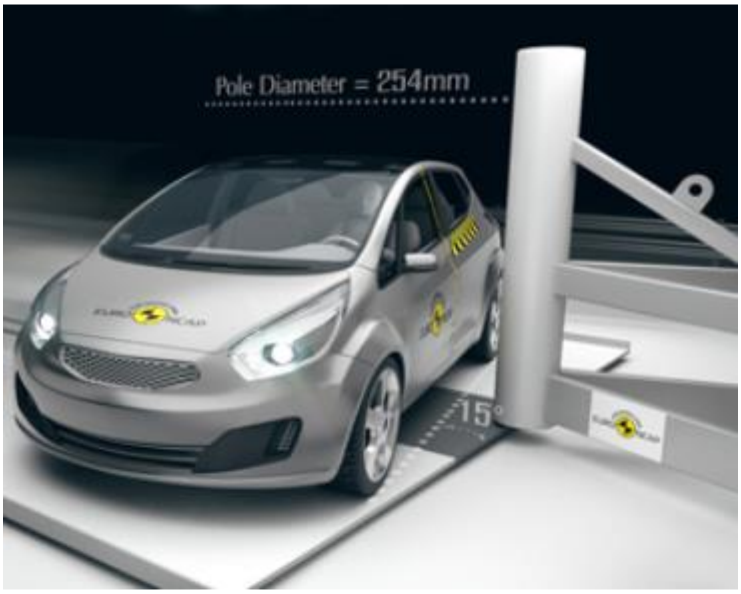
▲ Door Impact Model

Various spot welds are modeled in this example and we will create only the SW UP part of the five sets through exercises. In the model tree, the part names of spot welds are nomenclature as SW12, SW15, etc. This means that SW12 means that parts 1 and 2 are welded with spot welds.



▲ Spot Welding Part

The picture below shows the experimental test setup of the side impact of the vehicle column against the actual door. It is one of the important items in the collision safety test of automobiles. The impact condition of 32km/h speed on a 25cm column is tested to evaluate the deformation and penetration depth into the passenger seat.



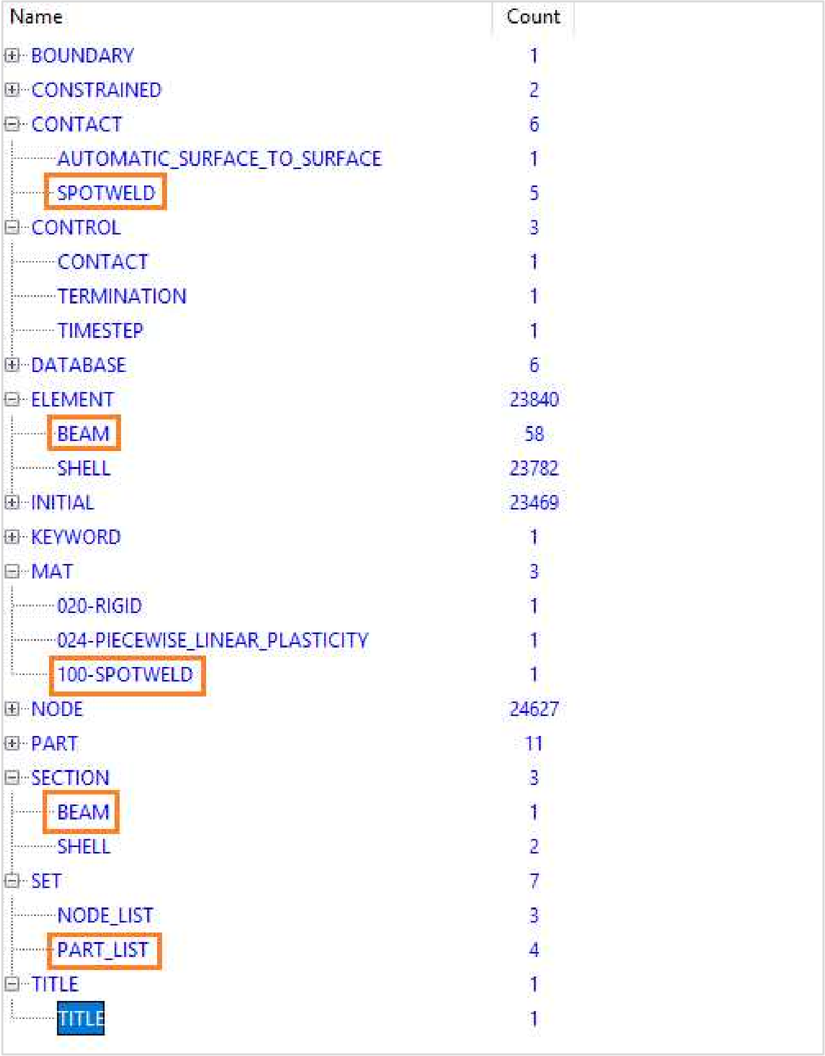
Side Pole Collision Test of Door [10]

---

[10] Image credit : Euro NCAP side impact test of FIAT Punto, 2017

Keyword Structure of Spot Welding

Let's take a look at the structure of the keyword to create Spot Weld. First, create spot weld Beam elements using the Spot Weld Generation tool. Next, create **MAT100_SpotWeld** to set the property, also use set **Beam** element section property for Spot Weld and define each property and section property in Spot Weld created in Part. And moving on, at * **Contact_Spotweld**, set the **part_set** and spot weld part of the target panel to be welded. In **DATABASE_ASCII, SWFORC** card must be selected to extract load or deformation applied to spot weld.



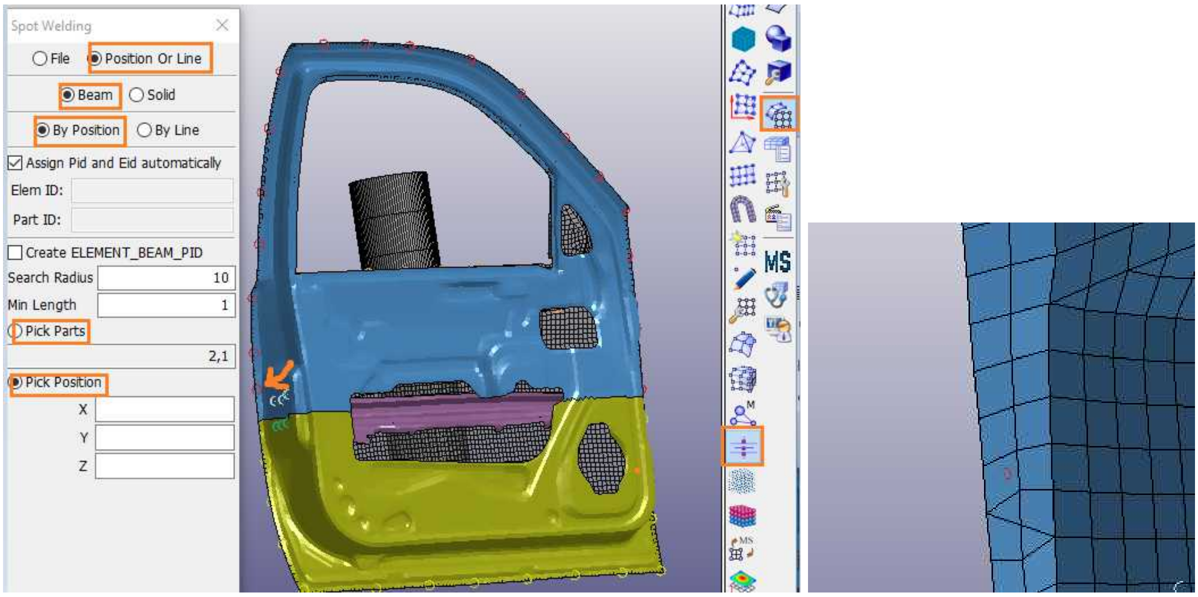
Name	Count
⊞ BOUNDARY	1
⊞ CONSTRAINED	2
⊟ CONTACT	6
AUTOMATIC_SURFACE_TO_SURFACE	1
SPOTWELD	5
⊟ CONTROL	3
CONTACT	1
TERMINATION	1
TIMESTEP	1
⊞ DATABASE	6
⊟ ELEMENT	23840
BEAM	58
SHELL	23782
⊞ INITIAL	23469
⊞ KEYWORD	1
⊟ MAT	3
020-RIGID	1
024-PIECEWISE_LINEAR_PLASTICITY	1
100-SPOTWELD	1
⊞ NODE	24627
⊞ PART	11
⊟ SECTION	3
BEAM	1
SHELL	2
⊟ SET	7
NODE_LIST	3
PART_LIST	4
⊟ TITLE	1
TITLE	1

Keyword Structure

Spot Welding Generation

Spot Weld beams or solid elements can be easily created by utilizing the Spot Weld option. In the case of ***Constrained_Spotweld**, it can be created only at the position of the node, while the Beam element can be created regardless of the position of the node as shown below. Spot weld function differs depending on the location of the beam element (center or near node). Generally, creating at the center of the element minimizes numerical errors. Let's set about 10 locations along the edge of the upper part of the door as shown. If you check the model tree after creation, you can see that a part consisting of one Beam element is created. If the number of spot welds is quite large, make a line along with the welding position in **Geometry** → **Curve**. You can also create spot welds all at once by entering a gap or the total weld number required on this line. However, if the shape is complex like a door, setting individual positions can be effective.

- **Element and Mesh → Spot Weld → Position | Beam | By Position → Pick Part = Select the part to be welded, click on the inner/outer panel of the upper door in the figure → Pick Position = select the position to spot weld**

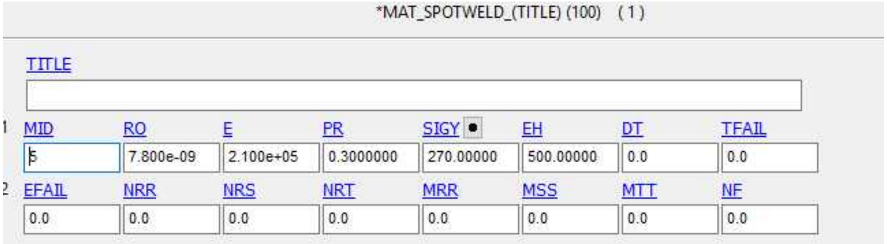


Spot Welding
○ File ● Position Or Line
● Beam ○ Solid
● By Position ○ By Line
☑ Assign Pid and Eid automatically
Elem ID:
Part ID:
☐ Create ELEMENT_BEAM_PID
Search Radius 10
Min Length 1
○ Pick Parts
2,1
● Pick Position
X
Y
Z

▲ Spot Weld Generation

Spot Welding Properties

The properties of Spot Weld use the **MAT100_SPOTWELD** keyword as explained in the previous description. The figure below shows the generated data. Actually, the property of spot weld should be able to measure the stiffness and strength through tensile tests or lap joint tests and then input its value. In general, the welded part has the characteristics of showing a partial increase in stiffness and hardness while breaking strength is somewhat lower than that of the base metal. In the Beam section characteristic, select ELFORM = 9 (Spotweld Beam), and select **QR** = 1 (Number of Integration). Select **CST** = 1 (Circular), which defines the cross-sectional area and the diameter is about 2mm because the panel to be welded is about 3 ~ 5mm thick.

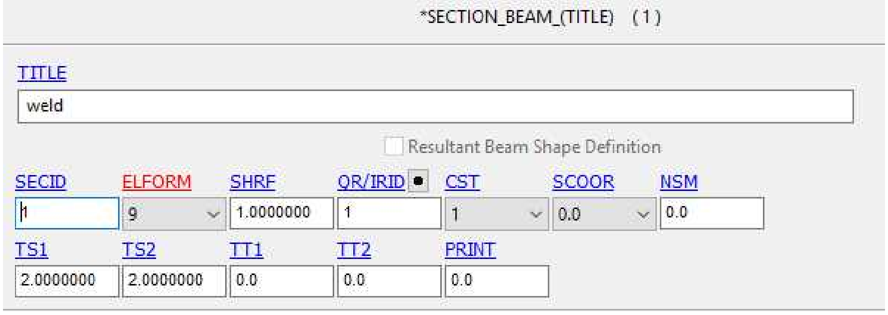


*MAT_SPOTWELD_(TITLE) (100) (1)

TITLE

1	MID	RO	E	PR	SIGY ●	EH	DT	TFAIL
	5	7.800e-09	2.100e+05	0.3000000	270.00000	500.00000	0.0	0.0
2	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

▲ MAT100_SpotWeld properties



*SECTION_BEAM_(TITLE) (1)

TITLE

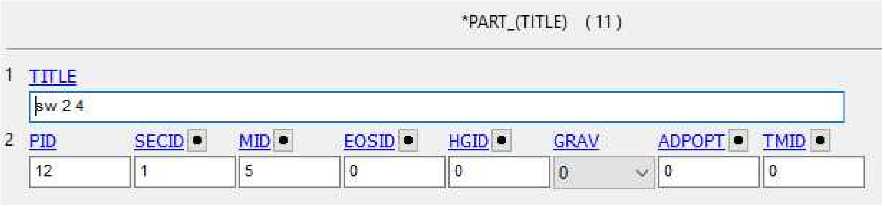
weld

☐ Resultant Beam Shape Definition

SECID	ELFORM	SHRF	QR/IRID ●	CST	SCOOR	NSM
1	9	1.0000000	1	1	0.0	0.0
TS1	TS2	TT1	TT2	PRINT		
2.0000000	2.0000000	0.0	0.0	0.0		

▲ Beam section properties for Spot Weld

In Part, assign the properties and section properties created to the spot weld part.



*PART_(TITLE) (11)

1 TITLE

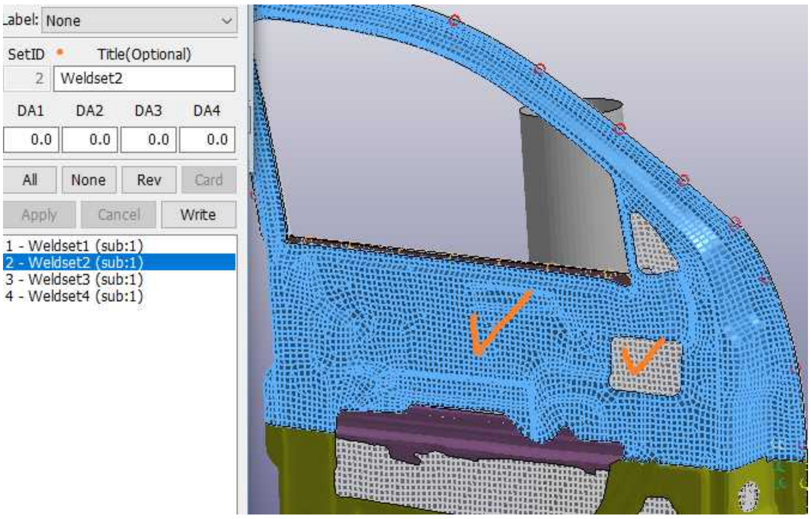
sw 2 4

2	PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
	12	1	5	0	0	0	0	0

▲ Part properties setup

Contact_Spotweld setup

Set the part to be joined by welding to one Part Set and then set the spot weld part and panel set in * **Contact_Spotweld**. Make a set by selecting the inner top panel and outer panel as shown below. Select the part set and spot weld parts created from **Contact_Spotweld**. If this process is not performed properly while modeling, you often experience error messages, so it is advisable to set it carefully.



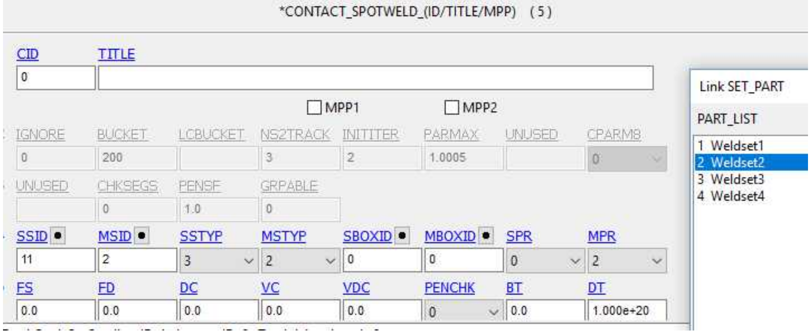
Label: None

SetID	Title(Optional)
2	Weldset2

DA1	DA2	DA3	DA4
0.0	0.0	0.0	0.0

1 - Weldset1 (sub:1)
2 - Weldset2 (sub:1)
3 - Weldset3 (sub:1)
4 - Weldset4 (sub:1)

▲ Generation of a Panel Set to be welded



*CONTACT_SPOTWELD_(ID/TITLE/MPP) (5)

CID	TITLE						
0							

☐ MPP1 ☐ MPP2

IGNORE	BUCKET	LCBUCKET	NS2TRACK	INITITER	PARMAX	UNUSED	CPARM8
0	200		3	2	1.0005		0
UNUSED	**CHKSEGS**	**PENSF**	**GRPABLE**				
	0	1.0	0				
SSID ●	**MSID** ●	**SSTYP**	**MSTYP**	**SBOXID** ●	**MBOXID** ●	**SPR**	**MPR**
11	2	3	2	0	0	0	2
FS	**FD**	**DC**	**VC**	**VDC**	**PENCHK**	**BT**	**DT**
0.0	0.0	0.0	0.0	0.0	0	0.0	1.000e+20

Link SET_PART

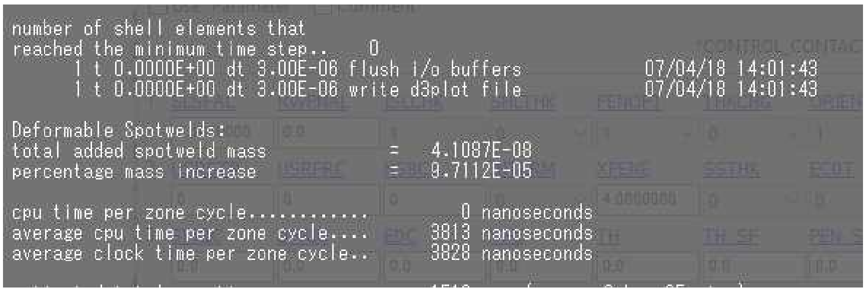
PART_LIST

1 Weldset1
2 Weldset2
3 Weldset3
4 Weldset4

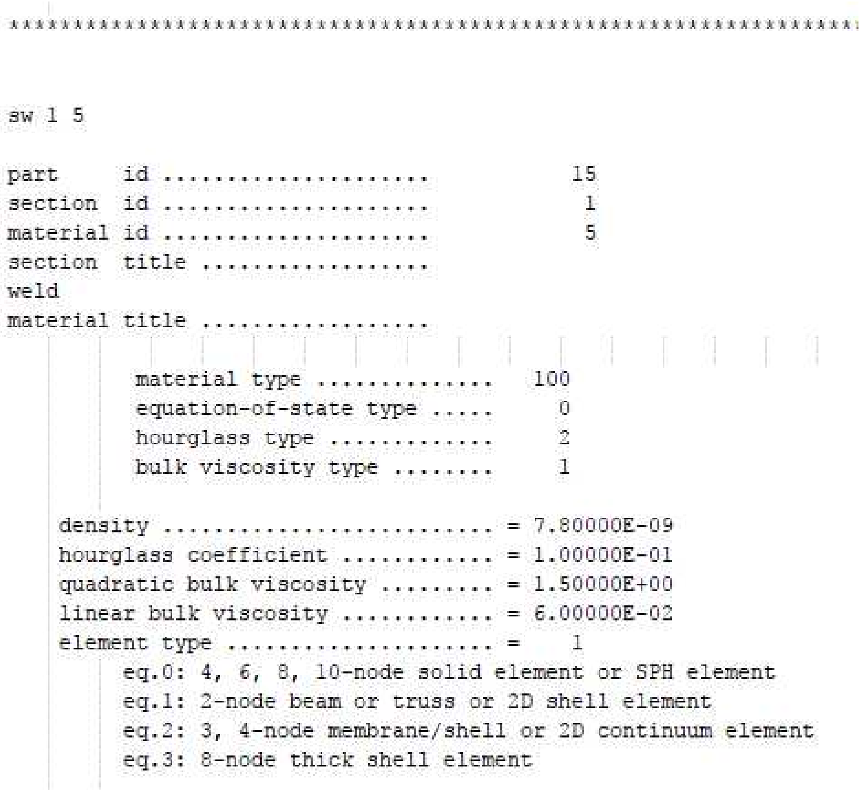
▲ *Contact Spot weld Setup

Analysis Condition

Basic analysis time, initial conditions and boundary conditions have been established. If you check the * **Control_Contact** keyword, you can see that **OUTSEG** = 1 is set. If **OUTSEG** = 1 is set, information related to welding is recorded in the history file, **D3HSP**, so it can be checked whether our modeling has been properly performed. If the spot weld is correctly modeled, the weight information of the spot weld is shown on the analysis log screen as shown below.



```
number of shell elements that
reached the minimum time step..    0
      1 t 0.0000E+00 dt 3.00E-06 flush i/o buffers            07/04/18 14:01:43
      1 t 0.0000E+00 dt 3.00E-06 write d3plot file            07/04/18 14:01:43

Deformable Spotwelds:
total added spotweld mass             =   4.1087E-08
percentage mass increase              =   9.7112E-05

cpu time per zone cycle............         0 nanoseconds
average cpu time per zone cycle....      3813 nanoseconds
average clock time per zone cycle..      3828 nanoseconds
```



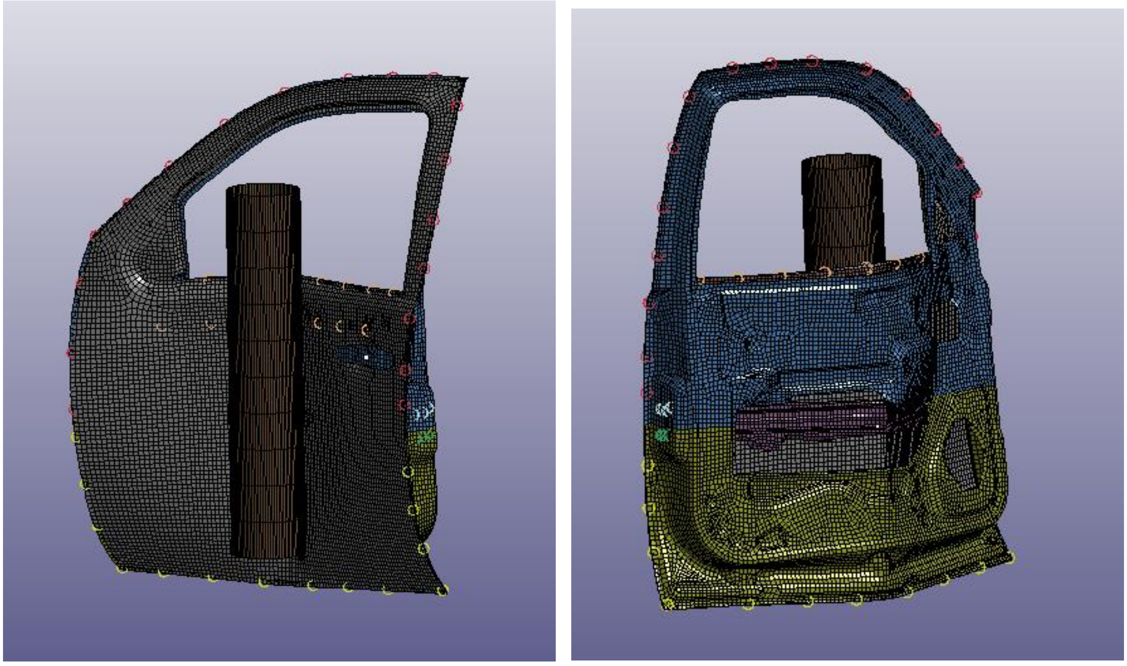
```
**********************************************************************


sw 1 5

part     id ....................         15
section  id ....................          1
material id ....................          5
section  title .................
weld
material title .................

          material type ..............   100
          equation-of-state type .....     0
          hourglass type .............     2
          bulk viscosity type ........     1

    density ......................... = 7.80000E-09
    hourglass coefficient ........... = 1.00000E-01
    quadratic bulk viscosity ........ = 1.50000E+00
    linear bulk viscosity ........... = 6.00000E-02
    element type .................... =     1
         eq.0: 4, 6, 8, 10-node solid element or SPH element
         eq.1: 2-node beam or truss or 2D shell element
         eq.2: 3, 4-node membrane/shell or 2D continuum element
         eq.3: 8-node thick shell element
```

▲ Spot Weld information recorded in the log screen and D3HSP

Now that you have created additional Spot Welds, let's perform an analysis. In the training model, Mass Scaling was partially applied to reduce analysis time. If you want to remove mass scaling, enter 0 on the **DT2MS** card of *** Control_Timestep**.

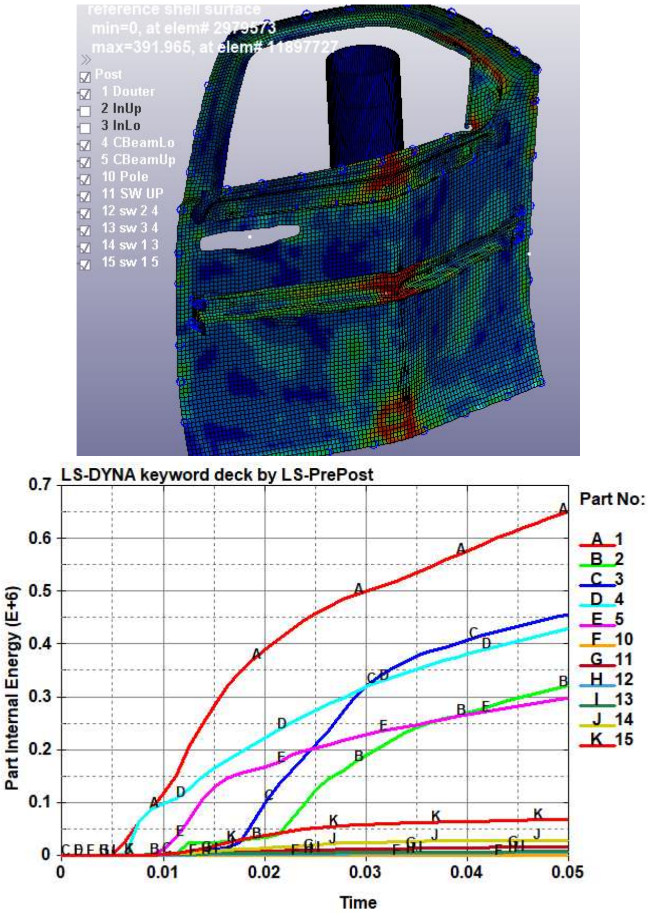
Analysis Result

The analysis will be solved within 30 minutes. The figure below shows the deformation pattern at the last frame of the analysis time of 0.05 seconds. On the result screen, you can see that the position of the weld is displayed as a circular mark.



▲ Deformation form in the last frame of side collision analysis

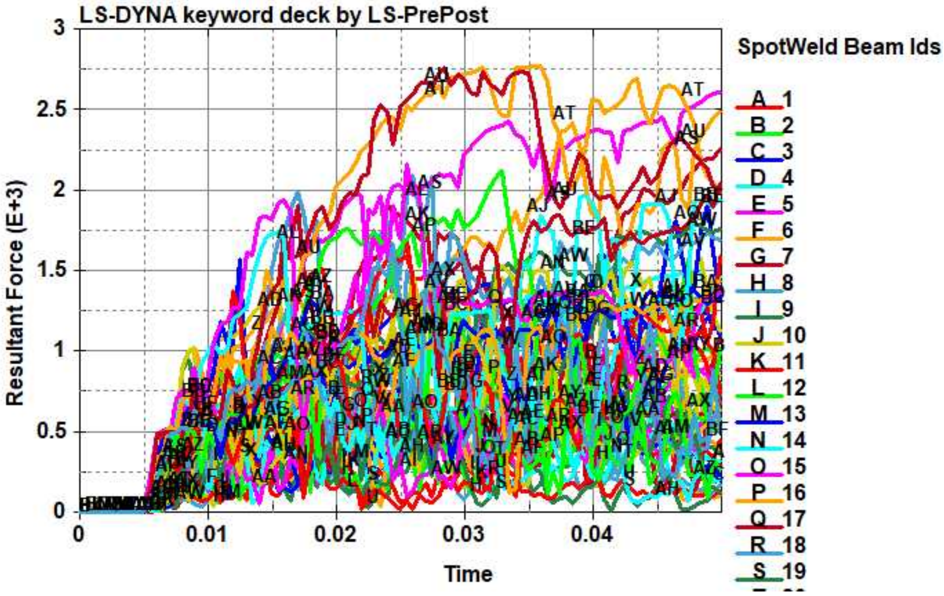
The figure below shows the Von Mises stress distribution. In the case of concentrated local shocks such as poles, large loads are concentrated on the support part of the impact bar, resulting in severe bending deformation. The key factor in the design of the impact bar is how effectively this constant weight can increase the bending resistance. The figure below right shows the internal energy distribution by component absorbed by the impact. Doors 1 ~ 5 are Door parts and 10 ~ 15 are Spot Weld parts.



▲ Von Mises Stress Distribution and Strain Energy Distribution by Component

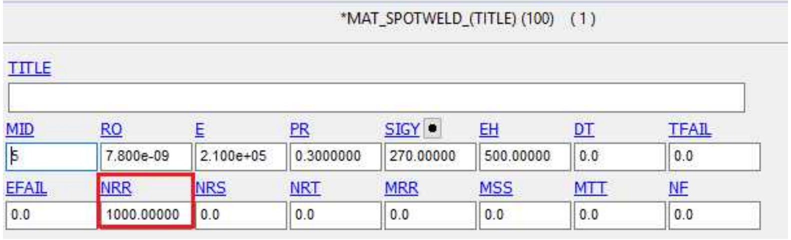
Spot Weld parts can also absorb some energy. In particular, if the number of spots absorb is increased by selecting the part where energy is absorbed a lot, the applied energy is distributed lower and it is possible to reduce the failure of the welding part.

Results file **ASCII → SWFORC | Load → All | Resultant Force → Plot** and check the load applied to the spot weld. The working load in the individual beams can be checked for each spot weld location. Likewise, increasing the weld number at the site of the highest load may be a solution for the enhancement of door structure integrity.



▲ Resultant Force Distribution of Working Load Extracted from Spot Weld

In this model, the failure condition of Spot Weld is applied to the load of 2.7kN or more, and it is analyzed that the failure criteria is set to 1kN based on Axial Force.



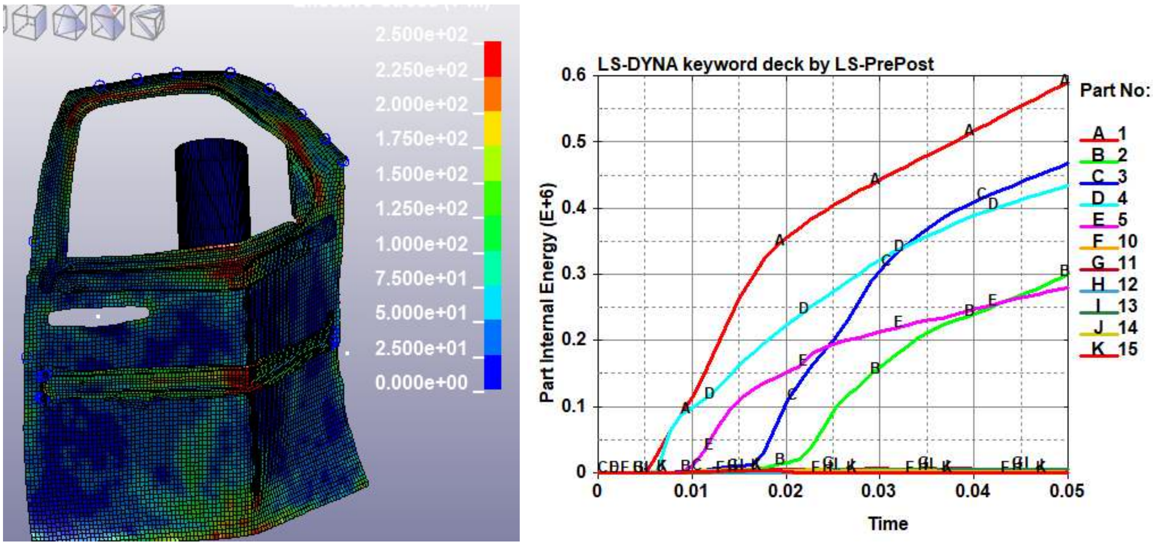
*MAT_SPOTWELD_(TITLE) (100) (1)

TITLE

MID	RO	E	PR	SIGY ●	EH	DT	TFAIL
5	7.800e-09	2.100e+05	0.3000000	270.00000	500.00000	0.0	0.0
EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	NF
0.0	1000.00000	0.0	0.0	0.0	0.0	0.0	0.0

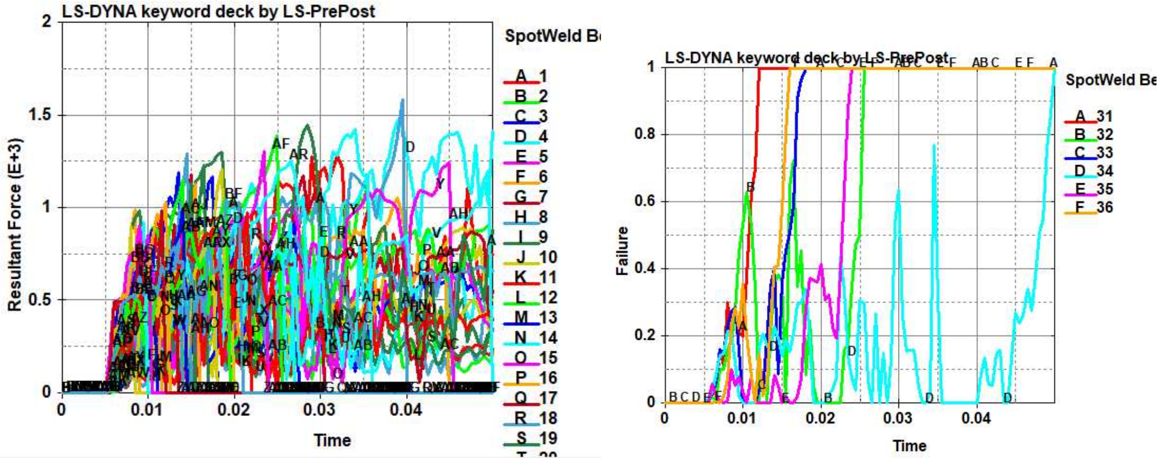
▲ *MAT100 Failure criterion NRR (Axial Force) setting

The figure below shows the distribution of Von Mises and the internal energy of the model to which Spot Weld's failure criterion is applied. Slight differences will be found with models that do not have failure criteria. In the spot welds of 10 ~ 15, failure occurred at the site where severe deformation occurred, and the energy did not increase. In the spot welded area, the stress was released, and the stress distribution and the size were slightly different.



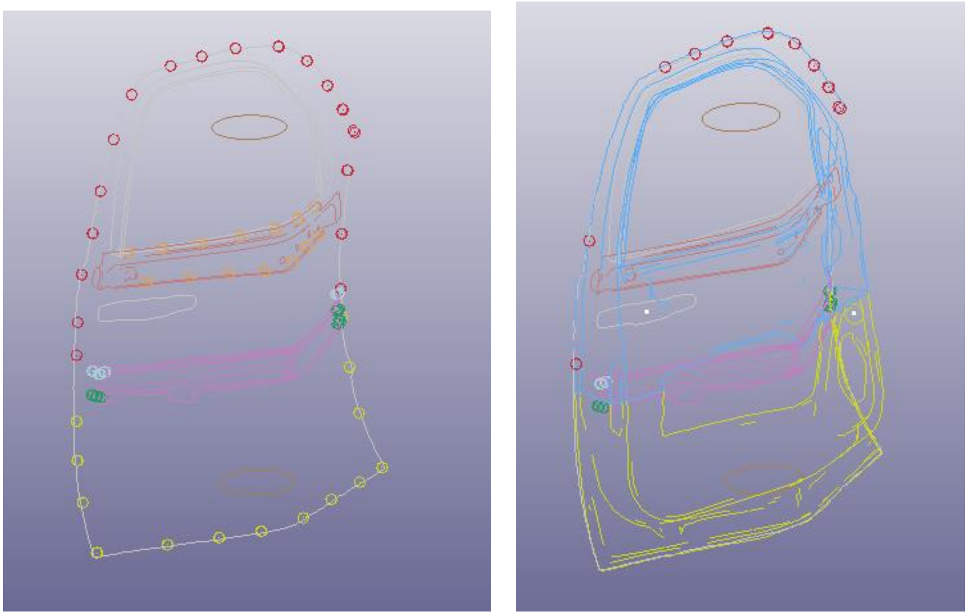
▲ Von Mises stress distribution and strain energy distribution by component

Plot the resultant forces and failures in the ASCII result file. In the resultant force, an area of more than 1 kN is calculated with rather high value due to the shear force or compressive load. On the other hand, it can be seen that all welds with an excess of 1 kN of axial force are failed (Failure index → 1).



▲ Resultant Force and Failure distribution

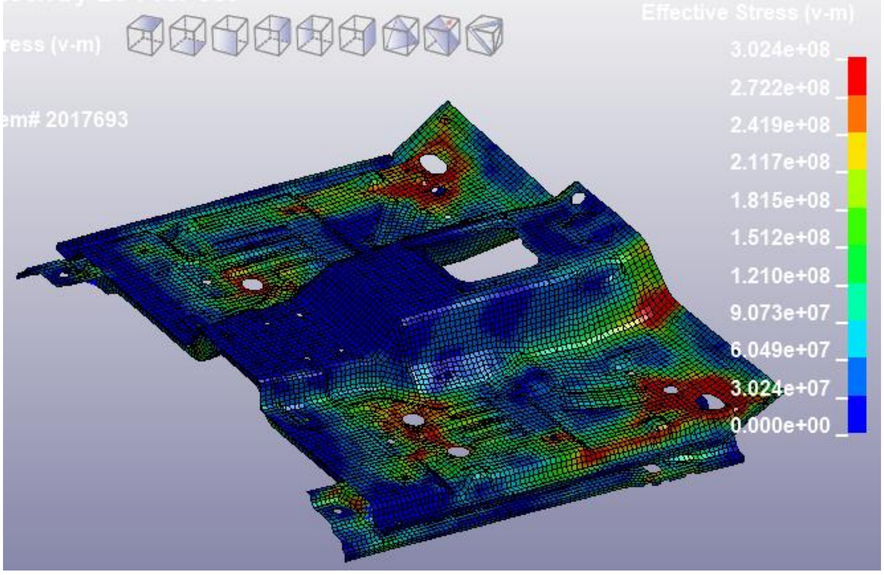
If the results are displayed in the **D3PLOT** in the **Feature Line Mode**, it is possible to visually check whether the Weld has broken at any point. The figure below shows the spot weld distribution in the last frame. The failure standard model can confirm that many of the welds are broken.



▲ Remained Spot Weld after impact (Left: No failure model, Right: Failure criteria applied model)

I hope that this course will give you an opportunity to learn about the modeling procedure, as well as conceptual understanding of spot welding modeling and its analysis.

Chapter 17. Floor Panel analysis with Restart



Workshop Objectives

✓ **Learn how to use Restart**
✓ **Practice Load Analysis of vehicle Floor Panel**
✓ **Practice how to create Small Restart keyword**

Workshop Duration: 40 min

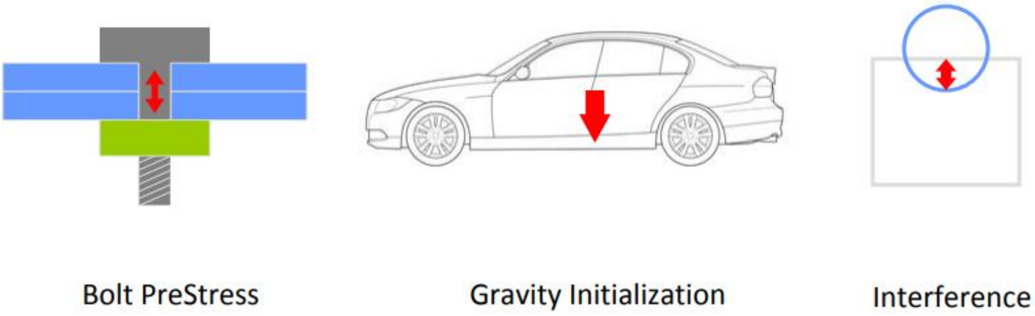
Application: Automotive, Restart, Implicit Load Analysis

Model Files

- W17_RS_FloorFrame.k: 1st analysis model with the initial load
- W17_RS_Start.k: Restart keyword with modified load

Introduction of Restart

In many analysis cases, it is often necessary to maintain the initial preloading state. For example, the application of various load cases under the condition of its own weight, the application of the static load under the pre-tension of the bolt, or the analysis of the load after the initial contact condition, for example, like to maintain the springback effect. In these cases, it is quite inefficient to analysis the initial condition repeatedly in every load case which is quite time-consuming. The technique of performing initial load conditions through separate analysis and applying various load cases or changing boundary conditions from the result obtained from the previous step is called **Restart analysis** technique.



Bolt PreStress

Gravity Initialization

Interference

▲ Restart Analysis Cases

In addition to the restart analysis, the effective analysis of these initial load conditions can be performed using the sequential method and dynamic relaxation method that applies the load sequentially (step-by-step analysis). In this exercise, we will practice how to use **Restart analysis method**. LS-DYNA provides three restart analysis methods:

- **Simple Restart**
- **Small Restart**
- **Full Restart**

Simple Restart simply stops the analysis in the middle and starts again from the point of interruption. In order to do that, simply enter r = **d3dump01** after the execution command.

```
>> lsdyna r=d3dump01
```

The second method, **Small Restart**, is a restart method that can modify only a part of the model. It is a restart method that can modify a limited part of the model such as the curve of the load, the partial modification of the contact condition and the partial modification of the element or node of the part. This method will be covered in this workshop which is a useful technique for applying various subsequent load cases under applying the same initial load condition. In other words, if the initial analysis proceeds to Time = 0 ~ 1sec, **Restart** analysis is applying various loads after the previous result from Time = 1sec. The initial and restart analysis conditions, i.e. precision (single or double) settings, number of CPUs and memory settings, must be the same for both the initial and restart analysis. In order to solve the restart, we need to create additional keywords for re-analysis. We will practice this part in detail. The execution method is to add **r=d3dump01** after the restart input file.

```
>> lsdyna i=restart.k r=d3dump01
```

The third method, **Full Restart**, can add new parts, add loads, boundary conditions and contact conditions for a completely new analysis to solve after a certain time. To execute, enter **n=d3full01** instead of r=d3dump01 in small restart.

```
>> lsdyna i=restart.k n=d3full01
```

Restart analysis requires files such as **d3dump**, **d3full,** and **runrsf** which are the binary files that store the results of the initial analysis. D3dumps and runrsf for small restarts are automatically created and can be requested by using *** DATABASE_BINARY_D3DUMP** or *** DATABASE_ BINARY_RUNRSF** when needed. There is also a command, in addition, if you press the switch key "**sw3**" after **control** + **C** during the analysis, you can get the **d3dump** file that stores the results up to that moment. As it is a very useful analysis technique for various load case analysis tasks with initial load conditions.

W17

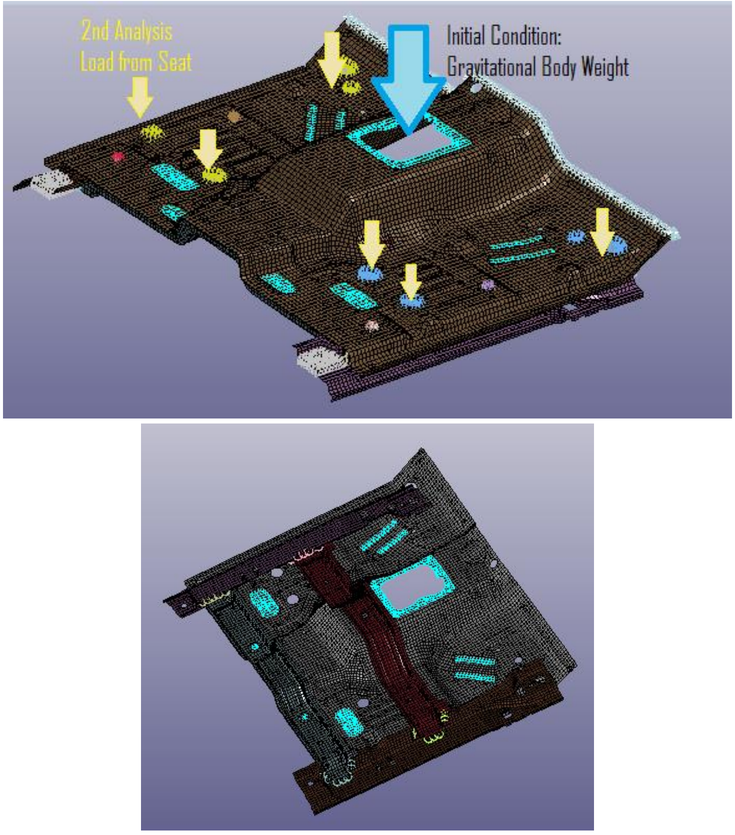
Workshop 17.

Floor Panel Load Analysis with Restart Method

Introduction of Restart

The restart analysis model is composed of a floor panel and frame of the vehicle. Let's import the ***W17_RS_FloorFrame.k*** model. A model consisting of five parts will appear as shown below. In this exercise, we will not practice the whole modeling methods, but only the **Restart** keyword generation part. Open the **Keyword Manager** of the model to see how the model is constructed. Two Cross Beams on the Floor Panel are welded by Spot Welding as practiced in Chapter 16. Both guide rails are bolted to the Floor Panel and are connected using **Constrained_CNRB**.

In the initial analysis, the body weight and the seat weight are applied to the entire model as shown below and the additional load factors on seat load that can be added during operation through restart are applied.

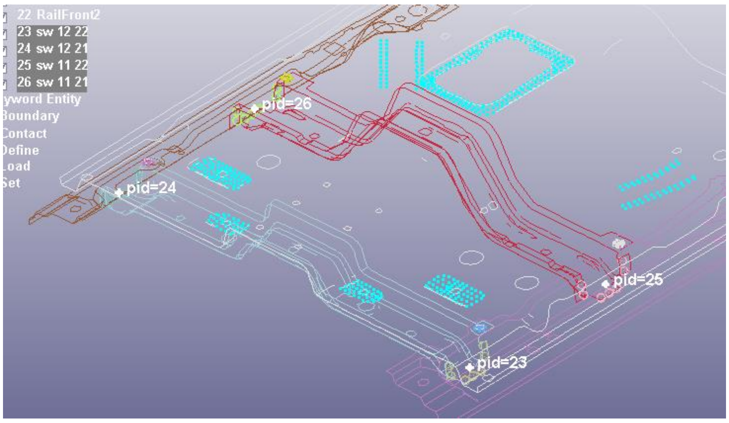


▲ Floor panel load analysis model

The initial analysis model for restart study is analyzed from 0-1 second and from 1 second to 2 seconds through invoking restart commands. Because static load analysis uses implicit solver, the concept of time does not mean much and the modeling procedure is briefly described as necessary.

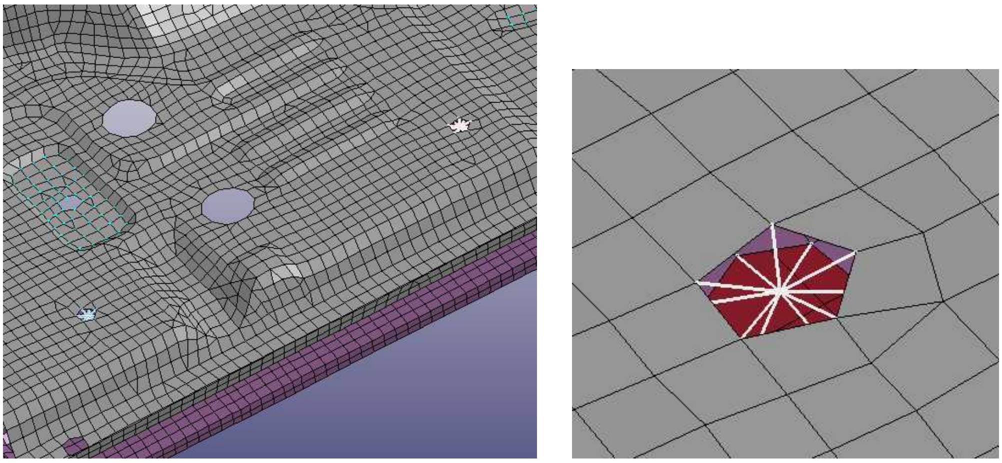
Initial Analysis

Two cross beams are welded to the guide rails on both sides with spot welds as shown below. The material property model used ***MAT100_Spotweld**.



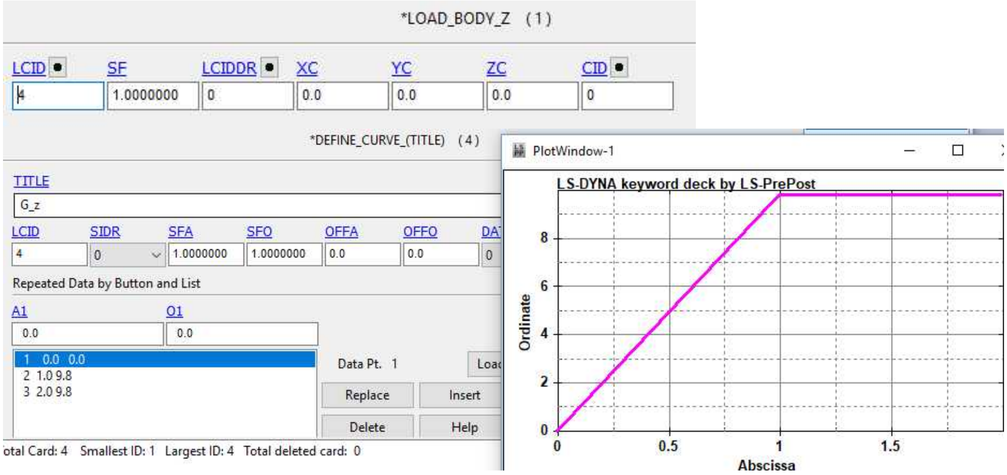
▲ Weldment of Cross Beam and Guide Rail

The bottom panel and guide rail were connected to each other using the ***Constrained_CNRB (Nodal Rigid Body)** which connects the nodes around the hole to be fastened as shown below. This is one of the very effective modeling methods for evoking a bolt fastening effect.



▲ Bolting location of the bottom panel and guide rail

For the applied load, *** Load_Body_Z** is applied so that its own weight load is applied to the whole model. Check the keyword creation screen and *** Define_Curve** shown below. And, vertical load 800N acting in Seat is applied as *** Load_Node_Set**. Check the location of the application and the size of the load in the figure below. Subsequent analysis through **Restart** will reset the Define Curve and proceed from T = 1.



*LOAD_BODY_Z (1)

LCID	SF	LCIDDR	XC	YC	ZC	CID
4	1.0000000	0	0.0	0.0	0.0	0

*DEFINE_CURVE_(TITLE) (4)

TITLE: G_z

LCID	SIDR	SFA	SFO	OFFA	OFFO	DAT
4	0	1.0000000	1.0000000	0.0	0.0	0

Repeated Data by Button and List

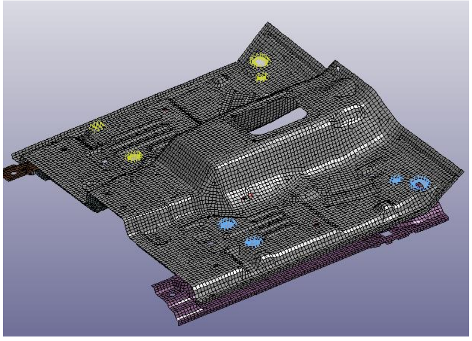
A1	O1
0.0	0.0

```
1  0.0  0.0
2  1.0 9.8
3  2.0 9.8
```

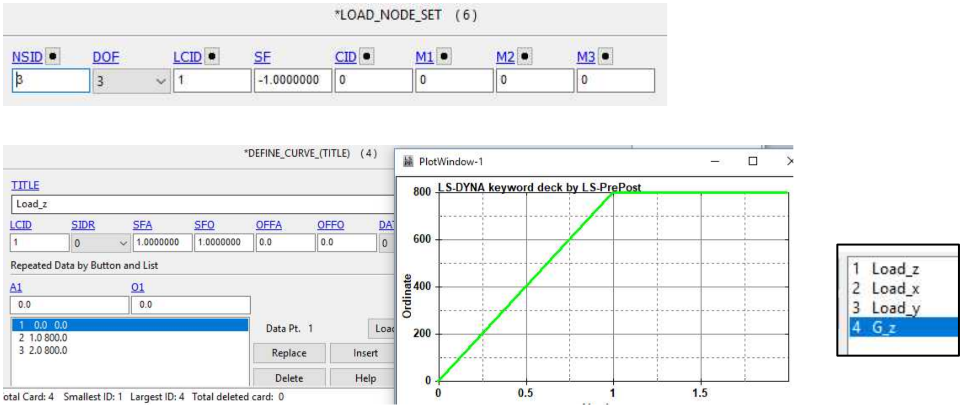
Data Pt. 1

otal Card: 4   Smallest ID: 1   Largest ID: 4   Total deleted card: 0

▲ Load_Body_Z setup for body weight



▲ Node_set for seat load



*LOAD_NODE_SET (6)

NSID	DOF	LCID	SF	CID	M1	M2	M3
3	3	1	-1.0000000	0	0	0	0

*DEFINE_CURVE_(TITLE) (4)

TITLE: Load_z

LCID	SIDR	SFA	SFO	OFFA	OFFO	DAT
1	0	1.0000000	1.0000000	0.0	0.0	0

Repeated Data by Button and List

A1	O1
0.0	0.0

```
1  0.0  0.0
2  1.0 800.0
3  2.0 800.0
```

Data Pt. 1

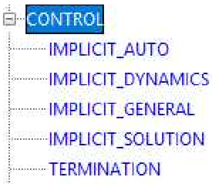
otal Card: 4   Smallest ID: 1   Largest ID: 4   Total deleted card: 0

```
1  Load_z
2  Load_x
3  Load_y
4  G_z
```

▲ Load_Node_Set with *Define_Curve for Seat Load

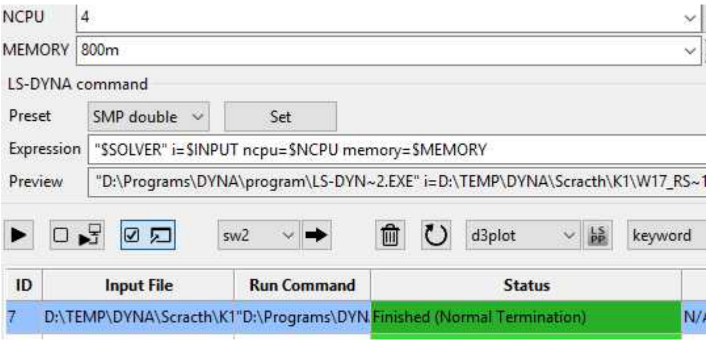
Analysis Setup

Since load analysis should be performed using an implicit scheme with the following * **Control** keywords. Open the keyword and check it once. See Appendix A5 for a description of the keywords used in Implicit analysis. (By default LS-DYNA uses Explicit solver to solve)



```
CONTROL
  IMPLICIT_AUTO
  IMPLICIT_DYNAMICS
  IMPLICIT_GENERAL
  IMPLICIT_SOLUTION
  TERMINATION
```

Let's perform an analysis of the initial model. Double precision must be applied to solve and setting of analysis execution command should be the same at Restart. Refer to the run command using LS-RUN. When the analysis is complete, you can see that **d3dump01** is created.



NCPU 4

MEMORY 800m

LS-DYNA command

Preset SMP double

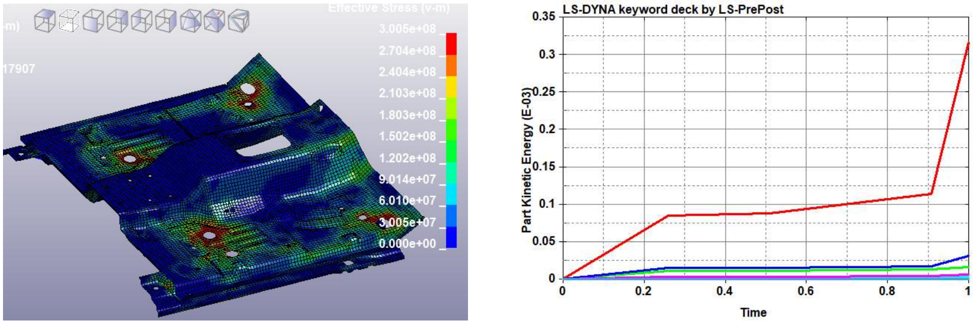
Expression "$SOLVER" i=$INPUT ncpu=$NCPU memory=$MEMORY

Preview "D:\Programs\DYNA\program\LS-DYN~2.EXE" i=D:\TEMP\DYNA\Scracth\K1\W17_RS~1

ID	Input File	Run Command	Status	
7	D:\TEMP\DYNA\Scracth\K1	"D:\Programs\DYN	Finished (Normal Termination)	N/

▲ Job execution of Initial analysis (LS-RUN)

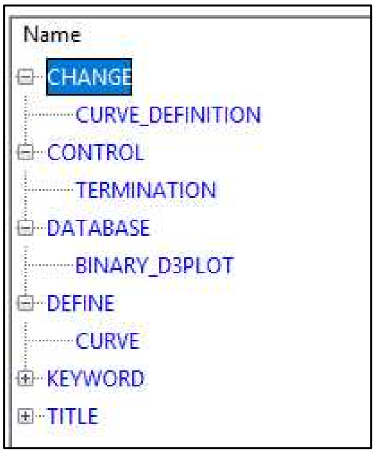
Observing at the analysis results, it can be seen that the self-weight and seat vertical load are applied up to 1 second so that the stress is concentrated on the corresponding area and the strain energy is generated as follows.



▲ Initial Analysis Results

Restart Keyword Generation

In the Restart analysis, the seat load will be applied in addition to the load applied in the initial analysis. Apply * **Change_Curve_Definition** keyword and * **Define_Curve_Title** keyword as follows. The restart keyword structure is as follows:

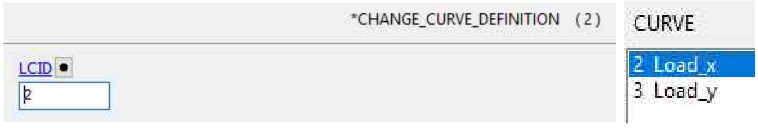


Name
- CHANGE
  - CURVE_DEFINITION
- CONTROL
  - TERMINATION
- DATABASE
  - BINARY_D3PLOT
- DEFINE
  - CURVE
- KEYWORD
- TITLE

▲ Restart Keyword structure

First, enter the curve to modify in the keyword * **Change_Curve_Definition**. At this time, the Curve ID should be the same as the initial condition and the point number inside the Curve should be the same. Termination time is 2 seconds.

- **Keyword Manager → Change_Curve_Definition → LCID → Curve 2 Load_X (repeat for Load_y)**



*CHANGE_CURVE_DEFINITION (2)

LCID ●

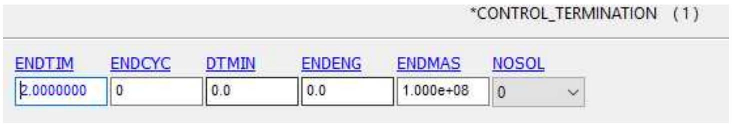
2

CURVE

2 Load_x

3 Load_y

- **Keyword Manager → Control_Termination → ENDTIM=2sec**

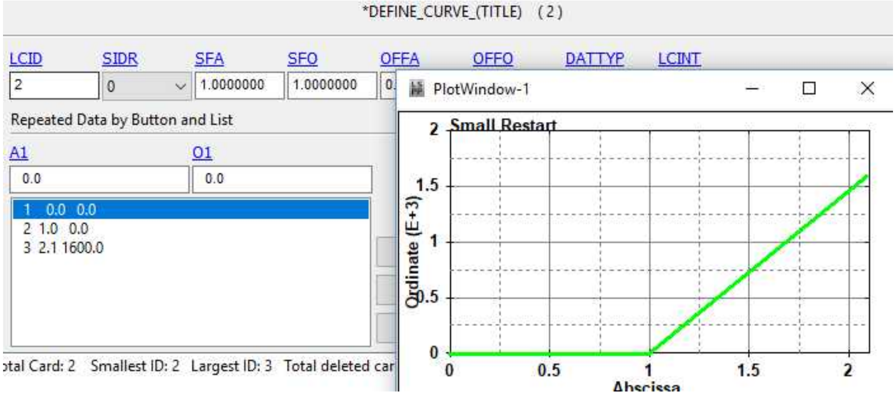


*CONTROL_TERMINATION (1)

ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS	NOSOL
2.0000000	0	0.0	0.0	1.000e+08	0

Load_x and **Load_y** are created to impart 1600N and 1200N after 1 second as follows.

- **Keyword Manager → Define_Curve → LCID=2, input A1, Q1(see image, and repeat for Load_y)**



*DEFINE_CURVE_(TITLE) (2)

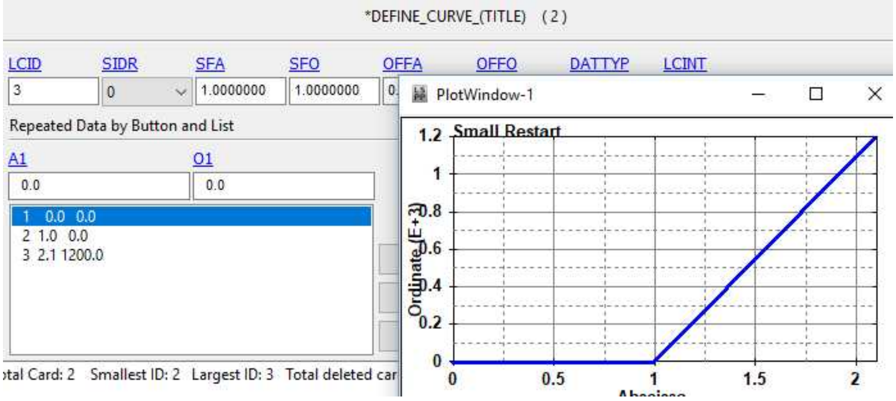
LCID	SIDR	SFA	SFO	OFFA
2	0	1.0000000	1.0000000	0.

Repeated Data by Button and List

A1	O1
0.0	0.0

1 0.0 0.0
2 1.0 0.0
3 2.1 1600.0

otal Card: 2 Smallest ID: 2 Largest ID: 3 Total deleted car



*DEFINE_CURVE_(TITLE) (2)

LCID	SIDR	SFA	SFO	OFFA
3	0	1.0000000	1.0000000	0.

Repeated Data by Button and List

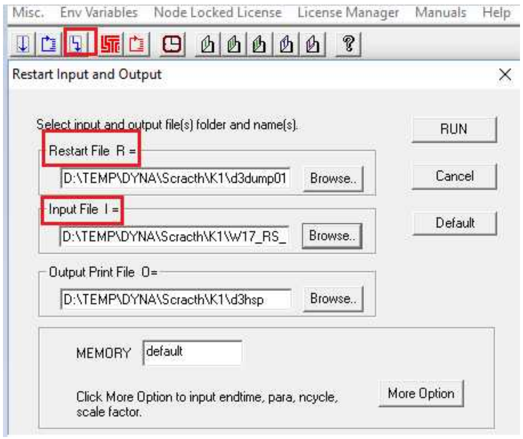
A1	O1
0.0	0.0

1 0.0 0.0
2 1.0 0.0
3 2.1 1200.0

otal Card: 2 Smallest ID: 2 Largest ID: 3 Total deleted car

▲ Load Curve Modification

Save the generated keyword as ***W17_RS_start.k*** and execute the calculation. The execution command should set the same conditions as the initial analysis. After the command, enter **i=W17_RS_start.k r = d3dump01**. When using LS-DYNA manager, select dump file and restart file as below. If you look at the working folder during the analysis, it was created earlier. It can be seen that **D3plot02** is generated following **D3plot01**.



Restart Input and Output

Select input and output file(s) folder and name(s).

Restart File R = D:\TEMP\DYNA\Scracth\K1\d3dump01

Input File I = D:\TEMP\DYNA\Scracth\K1\W17_RS_

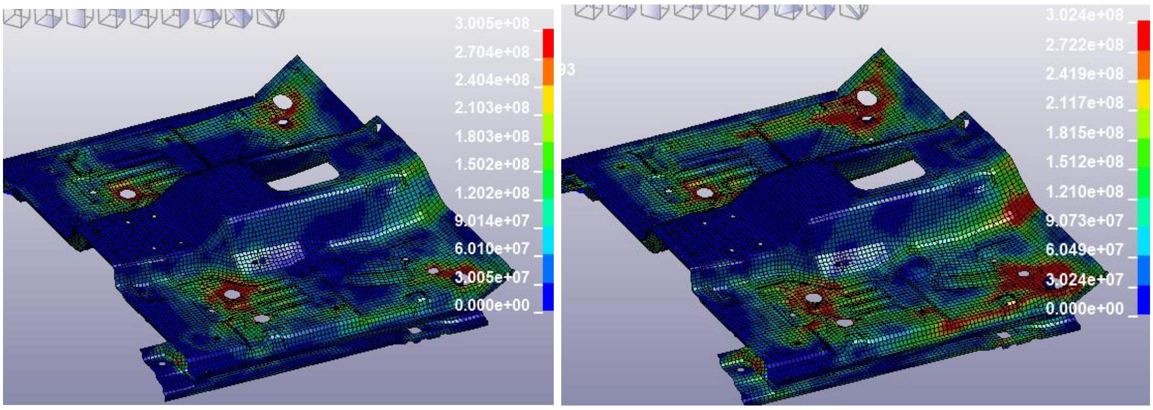
Output Print File O= D:\TEMP\DYNA\Scracth\K1\d3hsp

MEMORY default

Click More Option to input endtime, para, ncycle, scale factor.

Restart Analysis Result

If you open the d3plot, you can see that the frame of the analysis result has been added from t = 1 second to 2 seconds after the initial analysis. It can be seen that the stress acting due to Load_x and Load_y added during restart analysis are increased.



▲ Von mises stress distribution results (left: initial analysis, right: restart analysis)

So far, we have practiced the restart analysis procedure in this workshop. In addition to **Body_Load_z** that worked in the initial analysis, you should practice adding **Body_Load_y** and **Body_Load_x** during restart analysis to master the analysis process.

APPENDIX

Useful Tips and Know-How

Scan me

Models used in Workshops can be downloaded by scanning the QR code. The appendix summarizes tips and know-how that may be useful when conducting LS-DYNA analysis and research.

A1

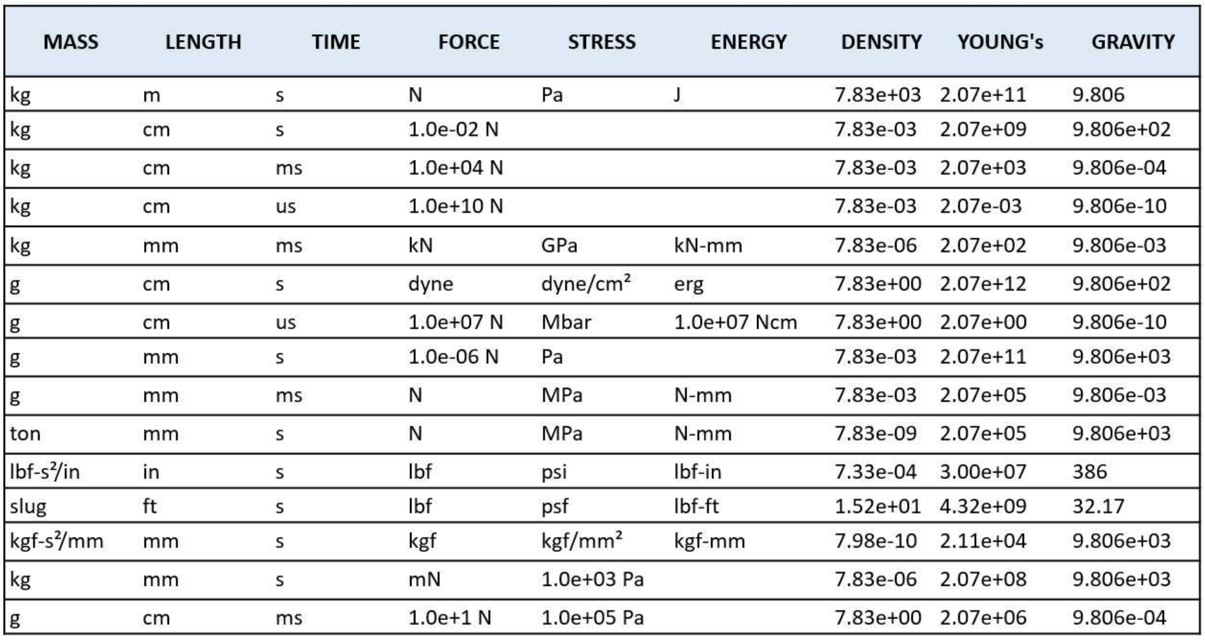
A1. Unit System

LS-DYNA is a very powerful explicit program. In the Explicit field, in particular, it has significant advantages over other larger programs. Although user convenience and user interface may seem like pre-aged old programs, they are still widely used in many industries. What is the most important thing about using this powerful analysis? This is the Unit system, which is the basis of the basics not only for LS-DYNA but for all other numerical program users. Matching units is the most important process in numerical analysis. If you spend a lot of time organizing complex shape modeling and analysis model setup and then, making tiny mistakes in the unit system, it leads to a total waste of time. In fact, you will experience such mistakes sometimes often during the initial career stage.

Historically, the biggest example of unit error was the loss of $ 1.25 million of Mars' satellites from NASA in 1999. Lockheed engineers in the United States used inch-based English units, while research institutes in other countries had problems using the metric system. Recently, as the multiphysics-based analysis is widely used, unit systems of physical quantities in multiple fields become more and more complicated and error occurrence in data compatibility and modeling between programs is expanding. As well as on how to reduce these mistakes, you can look at the results of the analysis and make intuitive judgments based on your experience. This chapter summarizes the unit system matching that can be used as a checklist. For reference, ***http://my.matereality.com/*** provides a free tool for matching unit systems by analysis program. There is also a dedicated tool for LS-DYNA.

LS-DYNA Unit system

The following table summarizes the property values of the Steel series for the unit system mainly used in LS-DYNA.

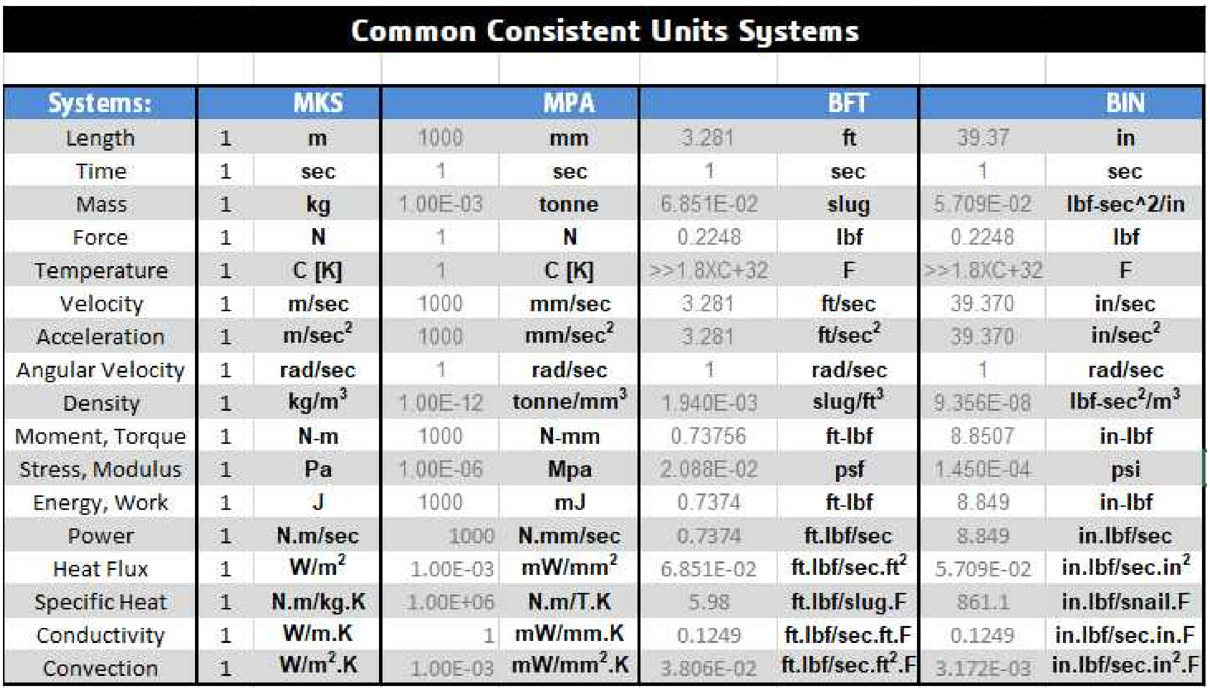


MASS	LENGTH	TIME	FORCE	STRESS	ENERGY	DENSITY	YOUNG's	GRAVITY
kg	m	s	N	Pa	J	7.83e+03	2.07e+11	9.806
kg	cm	s	1.0e-02 N			7.83e-03	2.07e+09	9.806e+02
kg	cm	ms	1.0e+04 N			7.83e-03	2.07e+03	9.806e-04
kg	cm	us	1.0e+10 N			7.83e-03	2.07e-03	9.806e-10
kg	mm	ms	kN	GPa	kN-mm	7.83e-06	2.07e+02	9.806e-03
g	cm	s	dyne	dyne/cm^2	erg	7.83e+00	2.07e+12	9.806e+02
g	cm	us	1.0e+07 N	Mbar	1.0e+07 Ncm	7.83e+00	2.07e+00	9.806e-10
g	mm	s	1.0e-06 N	Pa		7.83e-03	2.07e+11	9.806e+03
g	mm	ms	N	MPa	N-mm	7.83e-03	2.07e+05	9.806e-03
ton	mm	s	N	MPa	N-mm	7.83e-09	2.07e+05	9.806e+03
lbf-s^2/in	in	s	lbf	psi	lbf-in	7.33e-04	3.00e+07	386
slug	ft	s	lbf	psf	lbf-ft	1.52e+01	4.32e+09	32.17
kgf-s^2/mm	mm	s	kgf	kgf/mm^2	kgf-mm	7.98e-10	2.11e+04	9.806e+03
kg	mm	s	mN	1.0e+03 Pa		7.83e-06	2.07e+08	9.806e+03
g	cm	ms	1.0e+1 N	1.0e+05 Pa		7.83e+00	2.07e+06	9.806e-04

How to reduce mistakes in the unit system

1. Keep the unit system reference table handy when modeling.

One of the effective tips to prevent errors is to keep the unit system reference close to a reference while simulating multiphysics analysis. In particular, Europe and Asia mainly use the metric system, but in the United States, the imperial system of inch-lb.-sec or ft-lb-sec is used. Mixing these two unit systems can cause considerable confusion. In this case, be sure to refer to the unit system reference table for modeling. There are several reference tables in the chart, but again as mentioned mixing them can cause confusion. In general, it is recommended to select one of the following four reference tables (**MKS, MPA, BFT, BIN**) in consideration of the size of the model. If possible, the use of a metric unit system would minimize confusion in physical data compatibility if it is linked to fluid analysis or thermal analysis.



Common Consistent Units Systems

Systems:		MKS		MPA		BFT		BIN
Length	1	m	1000	mm	3.281	ft	39.37	in
Time	1	sec	1	sec	1	sec	1	sec
Mass	1	kg	1.00E-03	tonne	6.851E-02	slug	5.709E-02	lbf-sec^2/in
Force	1	N	1	N	0.2248	lbf	0.2248	lbf
Temperature	1	C [K]	1	C [K]	>>1.8XC+32	F	>>1.8XC+32	F
Velocity	1	m/sec	1000	mm/sec	3.281	ft/sec	39.370	in/sec
Acceleration	1	m/sec^2	1000	mm/sec^2	3.281	ft/sec^2	39.370	in/sec^2
Angular Velocity	1	rad/sec	1	rad/sec	1	rad/sec	1	rad/sec
Density	1	kg/m^3	1.00E-12	tonne/mm^3	1.940E-03	slug/ft^3	9.356E-08	lbf-sec^2/m^3
Moment, Torque	1	N-m	1000	N-mm	0.73756	ft-lbf	8.8507	in-lbf
Stress, Modulus	1	Pa	1.00E-06	Mpa	2.088E-02	psf	1.450E-04	psi
Energy, Work	1	J	1000	mJ	0.7374	ft-lbf	8.849	in-lbf
Power	1	N.m/sec	1000	N.mm/sec	0.7374	ft.lbf/sec	8.849	in.lbf/sec
Heat Flux	1	W/m^2	1.00E-03	mW/mm^2	6.851E-02	ft.lbf/sec.ft^2	5.709E-02	in.lbf/sec.in^2
Specific Heat	1	N.m/kg.K	1.00E+06	N.m/T.K	5.98	ft.lbf/slug.F	861.1	in.lbf/snail.F
Conductivity	1	W/m.K	1	mW/mm.K	0.1249	ft.lbf/sec.ft.F	0.1249	in.lbf/sec.in.F
Convection	1	W/m^2.K	1.00E-03	mW/mm^2.K	3.806E-02	ft.lbf/sec.ft^2.F	3.172E-03	in.lbf/sec.in^2.F

2. Intuitively judge input/output data based on one or two properties that can serve as criteria.

If you are an experienced engineer, it always better to memorize one or two properties in your head. For example, the most widely used materials are steel, and it has a density of about 7800 kg / m3, a modulus of elasticity of 210 GPa, a strength of about 800 MPa and compares the input and output results to find the abnormal values intuitively.

3. Be careful with Mass units in the US system.

One of the mistakes that occur in dynamics analysis is often experienced in defining mass when using the inch-pound-sec unit system. We are relatively familiar with units like kg and Newton through schooling. However, US units can be quite confusing in engineering. On most of the TV shows, pounds are often expressed as weight. But pounds are the unit of force (force, F = m • a) in engineering, exactly pound-force (lbf). The interpreter needs to know exactly what pounds mean in pound-mass (lbm) or pound-force (lbf). That is, 1 lbm = 0.4536 kg and 1 lbf equals 1 lbm to 32.174 lbm • ft/s^2 multiplied by gravitational acceleration 32.174 (ft/s^2). To eliminate this mistake, a metric system using kg and Newton, which is familiar with everyday life, would be a good alternative.

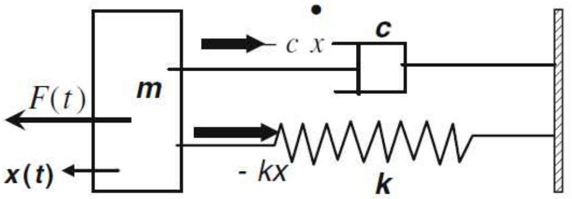
Many experienced engineers try to use the **MKS** [meter-kg-sec] system if possible unless it is a very small structure. If you use the MKS system in your head for a long time, you can intuitively check the results of the wrong results while analyzing the system. For example, a thermal analysis in the area of hundreds of degrees for a metal structure should be in the unit of displacement of 0.000X ~ 0.00XX. If it were in inch-lbm-sec units, who would not have been able to determine whether the level was adequate. If you're an engineer working primarily in the United States, things can be different. Readers will find a unit system that fits their work environment and make a familiar frame for the unit in their head. In particular, the interaction analysis of heat/fluid/electricity/structure, etc. through multiphysics makes the unit system turning out to be more complicated.

A2

A2. Time Step in Explicit Scheme

LS-DYNA is basically a non-linear Transient Dynamics FE code that can be explicit and implicit. To solve the problem of dynamics, we have to solve the following equation.

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F(t)$$



▲ SDOF spring-mass-damper system and governing equation

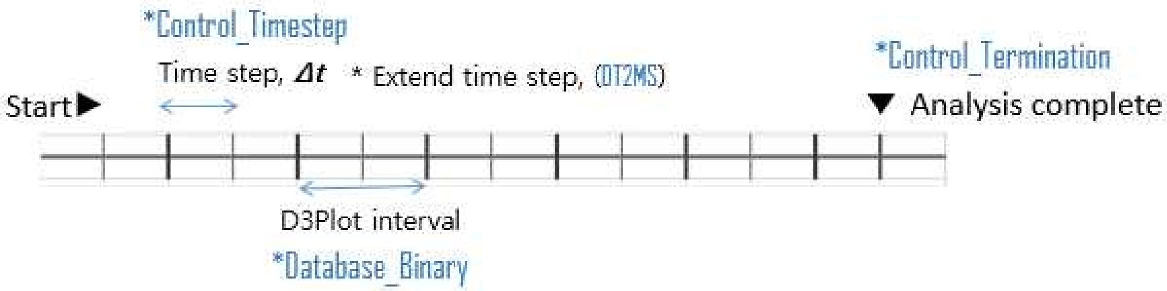
Where ***x* (t)** is a time step and ***kx* (t)** means internal force in the system. Here, if acceleration is defined as ***a***, velocity is ***v***, and distance is d and time step = ***n***, most of the problems to solve in LS-DYNA are to find d^{n+1} at t^{n+1}. In the case of using explicit code, the operation is done through the past n parameter known as the following formula, but in Implicit, we need to find the convergence point through iterative integration because we need to find the variable in $n+1$ situation that we do not know.

Explicit $\quad d^{n+1} = f(d^n, v^n, a^n, d^{n-1}, v^{n-1}, \ldots)$

Implicit $\quad d^{n+1} = f(v^{n+1}, a^{n+1}, d^n, v^n, \ldots .)$

If the nonlinearity is high, the explicit code will have a complete victory to solve the analysis until the termination time is reached. You might have learned from Chapter 7 about the difference between the models of interpretation between explicit and implicit. General-purpose programs such as ABAQUS and ANSYS often use implicit solvers and we can see that the analysis works normally even without entering the mass property. In Implicit, only the internal force balance fits well and iteration should be within the convergence range. Therefore, if only stiffness is reasonably input, the stiffness matrix can be used to derive the result by using the equilibrium relationship with the force. One of the advantages of implicit is that the analyst can define the time step size accordingly. Of course, the calculation of the stiffness matrix can take considerable analysis time depending on the equilibrium status of the system and the analysis can be completed utilizing a large step size in a very short time.

In the case of explicit, the nodal acceleration is calculated by dividing the nodal acceleration by the internal and external forces at each node. (Simply, F = m • a) The time step size used in the calculation is basically determined by the Courant condition algorithm. Not many engineers working on DYNA analysis fully understand the algorithms that determine the time step. This algorithm is basically determined by the size of the element and the speed of sound of the material and within the area that meets the criteria, the system deforms in a stable condition.



▲ Time Step Control Keyword

LS-DYNA can use both explicit and implicit codes. Explicit should be applied to phenomena such as collisions/impact phenomenon where dynamic deformation and analysis time are within microsec, and large deformations occur in materials with complex contact conditions and nonlinear properties. If implicit is applied, convergence does not occur and interpretation is impossible to solve or it takes a long time.

If you are studying engineering mathematics or numerical analysis, you can easily find the term CFL condition. Abbreviation of CFL is Courant-Friedrichs-Lewy Condition. This can be expressed as a convergence condition for the calculation of partial derivatives (PDE). The time step of LS-DYNA is obtained by this CFL condition. Sometimes expressed as a CFL length, an explicit and stable time step is defined as the time the wave speed passes through an element (i.e. stress wave propagation in the material).

$$C = \frac{u\,\Delta t}{\Delta x} \leq C_{\max}$$

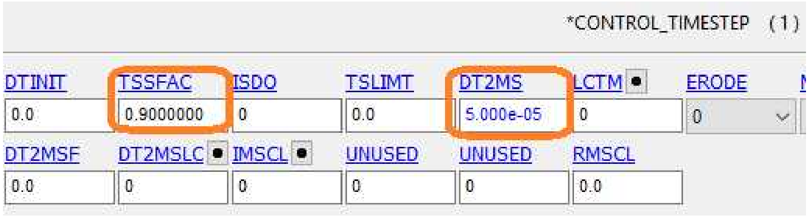
In the equation, Δx is the length of one element, u is the wave speed (speed obtained by Modulus and density) and Δt is the time step. Explicit is defined as C_{max} = 1 and **TSSFAC** variable corresponds to this Cmax value in * **Control_Timestep**. The default value of 0.9 is used to reduce the step size by reducing the **TSSFAC** to 0.5 to 0.6 for very soft materials (bird, rubber, fluids, etc.) or for very high nonlinear explosions.

Tips for using time steps when modeling

If you have an understanding of the time step concept in the explicit analysis, how can you effectively interpret and use it for analysis? As you may have noticed, the LS-DYNA analysis is a very time-consuming analysis. These workshop models are to carried out to deal with simple models, but the FE models in the real-world task can take from few minutes to as many days(example for long-duration run is while solving Engine Fan blade out analysis, it take more than two weeks for solving full-scale engine model). In this case, you can adjust the time step artificially by applying the following method to check the result quickly in the early stage of the modeling phase to see if the model runs without error.

- **Check the analysis result by applying mass scaling**

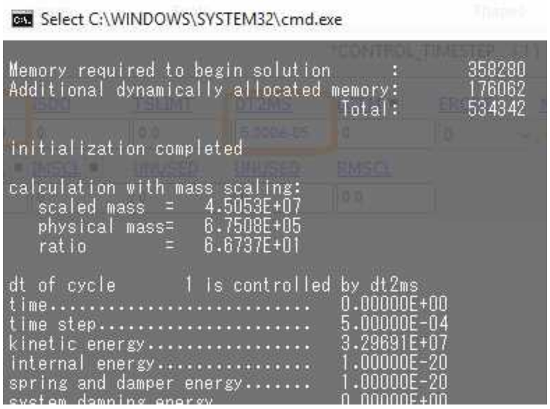
Enter the mass scaled time step value in **DT2MS** in ***Control_Timestep**.



*CONTROL_TIMESTEP (1)

DTINIT	TSSFAC	ISDO	TSLIMT	DT2MS	LCTM ●	ERODE	
0.0	0.9000000	0	0.0	5.000e-05	0	0	
DT2MSF	DT2MSLC ●	IMSCL ●	UNUSED	UNUSED	RMSCL		
0.0	0	0	0	0	0.0		

For example, in the case of the existing model, the time step size is 1e-5 sec, which is increased 5 times to 5e-5 in **DT2MS** and the analysis is executed. Depending on the analysis case, it is generally recommended that the mass scale value should be within **5%** of the total system and within **10%** for one part. The influence of the increased mass can be monitored in ***DATABASE_ASCII _GLSTAT and MATSUM.**



```
Memory required to begin solution      :      358280
Additional dynamically allocated memory:      176062
                                 Total:      534342

initialization completed

calculation with mass scaling:
   scaled mass  =   4.5053E+07
   physical mass=   6.7508E+05
   ratio        =   6.6737E+01

dt of cycle       1 is controlled by dt2ms
time...........................   0.00000E+00
time step......................   5.00000E-04
kinetic energy.................   3.29691E+07
internal energy................   1.00000E-20
spring and damper energy.......   1.00000E-20
system damping energy             0.00000E+00
```

▲ Log screen showing scaled mass data

- **Decrease the modulus value or increase the element size related to the CFL length**
- **Initial analysis after setting TSSFAC = 1**
- **Apply the original property model after checking the initial analysis result by setting the property to * MAT_Rigid.**

In this way, by intentionally adjusting the time step, it is possible to effectively reduce the analysis time for model checking during the initial model development stage.

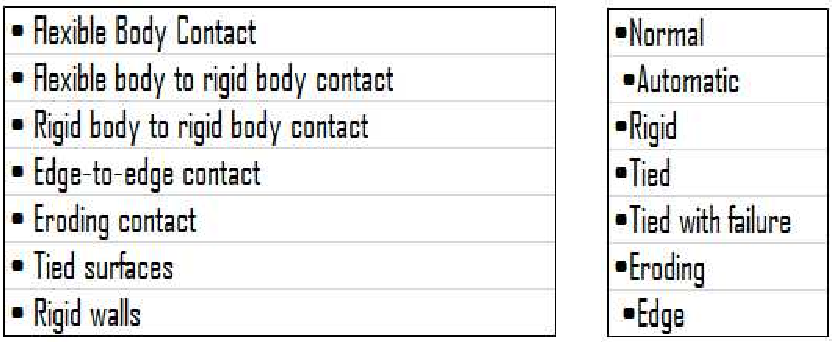
A3

A3. Contact Algorithms in LS-DYNA

One of the most key aspects of impact analysis is how similar contact is modeled in order to replicate real-world conditions, which in turn affects the reliability of the analysis. Nevertheless, one of many cases where many FE models fail is caused by inadequate contact models. However, not many engineers fully understand the theory and algorithms of contact theory. This chapter briefly explained the theoretical background of contact modeling, what contact models are and how to use them.

Contact Algorithm Types

LS-DYNA offers many contact conditions and which are quite complex options. For effective analysis, it is important to understand the characteristics of each contact model and option. The types and options of Contact algorithms provided by LS-DYNA are as follows.



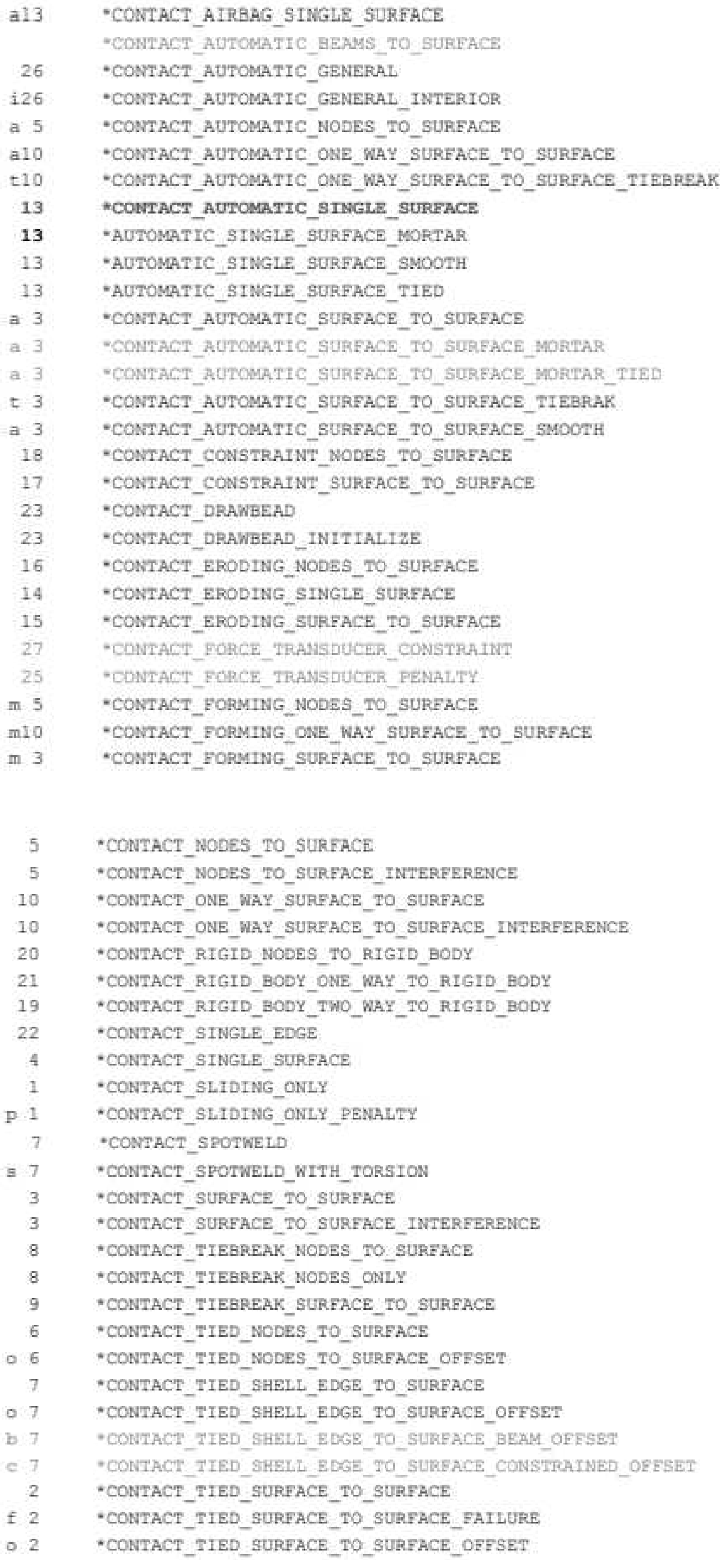
- Flexible Body Contact
- Flexible body to rigid body contact
- Rigid body to rigid body contact
- Edge-to-edge contact
- Eroding contact
- Tied surfaces
- Rigid walls

- Normal
- Automatic
- Rigid
- Tied
- Tied with failure
- Eroding
- Edge

▲ Contact Algorithm Types and Contact Options

Depending on the physical phenomenon in which contact occurs, it can be classified into five types. Let's take a closer look at the most frequently used Contacts while solving:

- **No deformation (Rigid wall)**
- **Sliding contact**
    - **3D contact(*Contact_option[1])**
    - **2D contact(*Contact_2D…)**
- **Contact considering shape (* Contact_entity)**
- **Cable contacts (* Contact_Guided_Cable)**
- **Internal contact of the solid element (* Contact_Interior)**

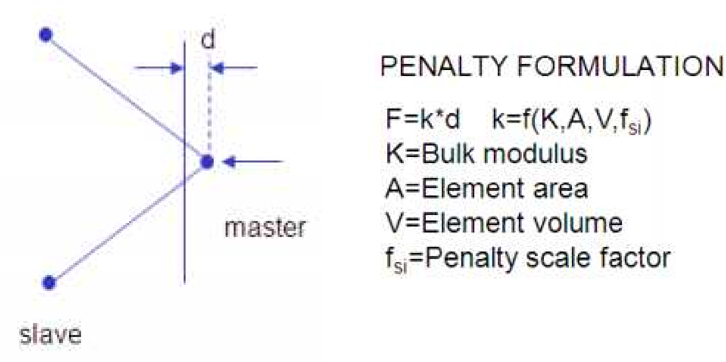


a13	*CONTACT_AIRBAG_SINGLE_SURFACE
	*CONTACT_AUTOMATIC_BEAMS_TO_SURFACE
26	*CONTACT_AUTOMATIC_GENERAL
i26	*CONTACT_AUTOMATIC_GENERAL_INTERIOR
a 5	*CONTACT_AUTOMATIC_NODES_TO_SURFACE
a10	*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
t10	*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
13	***CONTACT_AUTOMATIC_SINGLE_SURFACE**
13	*AUTOMATIC_SINGLE_SURFACE_MORTAR
13	*AUTOMATIC_SINGLE_SURFACE_SMOOTH
13	*AUTOMATIC_SINGLE_SURFACE_TIED
a 3	*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE
a 3	*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR
a 3	*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED
t 3	*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBRAK
a 3	*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH
18	*CONTACT_CONSTRAINT_NODES_TO_SURFACE
17	*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE
23	*CONTACT_DRAWBEAD
23	*CONTACT_DRAWBEAD_INITIALIZE
16	*CONTACT_ERODING_NODES_TO_SURFACE
14	*CONTACT_ERODING_SINGLE_SURFACE
15	*CONTACT_ERODING_SURFACE_TO_SURFACE
27	*CONTACT_FORCE_TRANSDUCER_CONSTRAINT
25	*CONTACT_FORCE_TRANSDUCER_PENALTY
m 5	*CONTACT_FORMING_NODES_TO_SURFACE
m10	*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE
m 3	*CONTACT_FORMING_SURFACE_TO_SURFACE
5	*CONTACT_NODES_TO_SURFACE
5	*CONTACT_NODES_TO_SURFACE_INTERFERENCE
10	*CONTACT_ONE_WAY_SURFACE_TO_SURFACE
10	*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE
20	*CONTACT_RIGID_NODES_TO_RIGID_BODY
21	*CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY
19	*CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY
22	*CONTACT_SINGLE_EDGE
4	*CONTACT_SINGLE_SURFACE
1	*CONTACT_SLIDING_ONLY
p 1	*CONTACT_SLIDING_ONLY_PENALTY
7	*CONTACT_SPOTWELD
s 7	*CONTACT_SPOTWELD_WITH_TORSION
3	*CONTACT_SURFACE_TO_SURFACE
3	*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE
8	*CONTACT_TIEBREAK_NODES_TO_SURFACE
8	*CONTACT_TIEBREAK_NODES_ONLY
9	*CONTACT_TIEBREAK_SURFACE_TO_SURFACE
6	*CONTACT_TIED_NODES_TO_SURFACE
o 6	*CONTACT_TIED_NODES_TO_SURFACE_OFFSET
7	*CONTACT_TIED_SHELL_EDGE_TO_SURFACE
o 7	*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFSET
b 7	*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET
c 7	*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET
2	*CONTACT_TIED_SURFACE_TO_SURFACE
f 2	*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE
o 2	*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET

▲ Contact models in LS-DYNA

Penalty Contact Theoretical Background

In LS-DYNA, contact and displacement are calculated basically by Penalty Contact Method. In the penalty method, the contact load is calculated through the relationship with the depth ***d*** penetrated by the penalty stiffness ***k***. Penalty stiffness is determined by the volume of the element, the area of contact and the stiffness of the element.



PENALTY FORMULATION

$F=k*d$ $k=f(K,A,V,f_{si})$
K=Bulk modulus
A=Element area
V=Element volume
f_{si}=Penalty scale factor

▲ Penalty formulation

The main disadvantage of the penalty method is that the contact behavior is governed by this penalty stiffness ***k***. If ***k*** is too large, high cycle vibrations occur and the explicit computational process becomes unstable. If ***k*** is too small, the depth of penetration becomes unrealistically large. To compensate for this drawback, **SOFT** option can be activated.

A common option is **SOFT** = 0 so that the contact stiffness is calculated from the element size and properties. In the case of shell and solid elements, the calculations are as follows.

- **Shell:** $k=slsfac\cdot sf\cdot K\cdot A/d$
- **Solid:** $k=slsfac\cdot sf\cdot K\cdot A^2/d$

Where slsfac : global scale factor (*Control_Contact)
sf : local scale factor (*Contact_ keyword(sfs, sfm))
K : Bulk modulus
A: element area
d: thickness

In the case of SOFT = 1, it is recommended to activate the 'Soft Constraint' and apply it in the case of rubber to iron impact, which is a contact state between a very soft material and a strong material. In the case of applying **SOFT** = 1, the contact stiffness ***k*** is expressed as the following formula and the larger value of the two ***k*** is used.

- **Shell:** $k_1 = slsfac \cdot sf \cdot K \cdot A/d$ $\quad$ $k_2 = 1/2 \cdot sofscl \cdot m_n/\Delta t^2$

  Solid: $k_1 = slsfac \cdot sf \cdot K \cdot A^2/d$ $\quad$ $k_2 = 1/2 \cdot sofscl \cdot m_n/\Delta t^2$

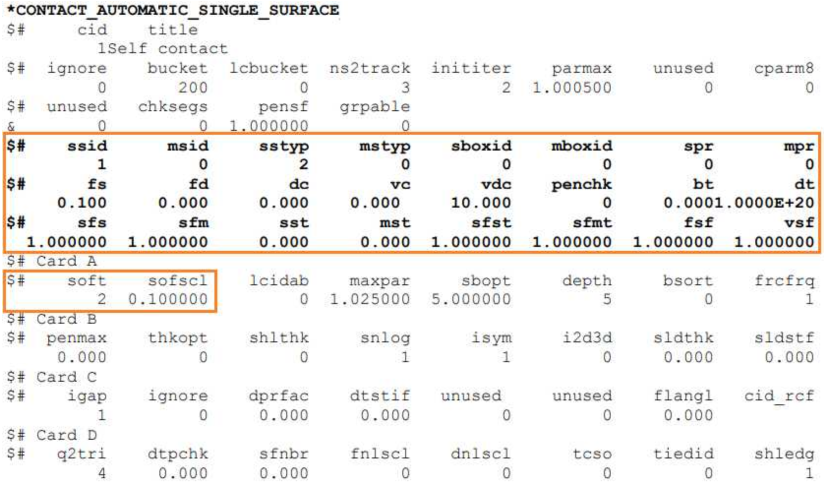
  $k = max(k_1, k_2)$

where Mn = nordal mass without mass scale
Δt=time step

Using **SOFT** = 2 of the contact options is a completely different way of calculating contact stiffness. This is calculated in the direction of the softer contact phenomenon. Unlike the previous methods, the contact load is calculated by passing through a segment of a specific region instead of penetrating through a specific segment. Therefore, if contact is unstable even if **SOFT** = 1 is applied, let's apply **SOFT** = 2 and compare the results. For most metal or composite collisions, applying **SOFT** = 0 or 1 would not be a problem.

$$k = slsfac \cdot sfs \cdot \frac{m1 \cdot m2}{(m1+m2)\Delta t_c^2}$$

Where m1,2=segement mass
Δtc=step size at t=c



```
*CONTACT_AUTOMATIC_SINGLE_SURFACE
$#     cid     title
         1Self contact
$#  ignore    bucket  lcbucket  ns2track  inititer    parmax    unused    cparm8
         0       200         0         3         2  1.000500         0         0
$#  unused   chksegs     pensf   grpable
&        0         0  1.000000         0
$#    ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
         1         0         2         0         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
     0.100     0.000     0.000     0.000    10.000         0     0.0001.0000E+20
$#     sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
  1.000000  1.000000     0.000     0.000  1.000000  1.000000  1.000000  1.000000
$# Card A
$#    soft    sofscl    lcidab    maxpar     sbopt     depth     bsort    frcfrq
         2  0.100000         0  1.025000  5.000000         5         0         1
$# Card B
$#  penmax    thkopt    shlthk     snlog      isym     i2d3d    sldthk    sldstf
     0.000         0         0         1         1         0     0.000     0.000
$# Card C
$#    igap    ignore    dprfac    dtstif    unused    unused    flangl   cid_rcf
         1         0     0.000     0.000         0         0     0.000
$# Card D
$#   q2tri    dtpchk     sfnbr    fnlscl    dnlscl      tcso    tiedid    shledg
         4     0.000     0.000         0         0         0         0         1
```

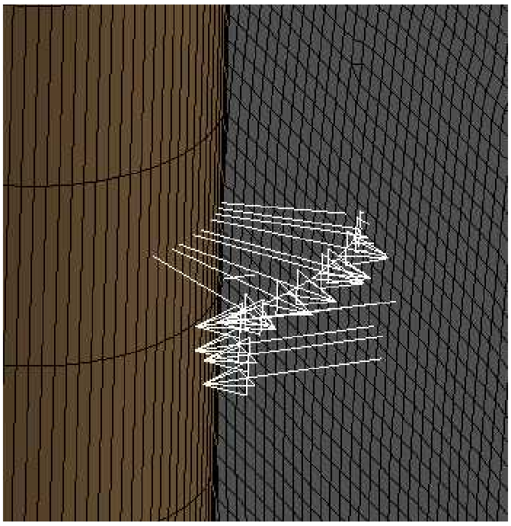
▲ Contact_automatic_single_surface Keyword example

Contact Types

• Automatic and Non-Automatic contact

In terms of contact conditions, there are **Automatic** conditions. In the case of a convention with an automatic card, bidirectional contact can occur regardless of the normal direction of the shell and the shell thickness is always included in the contact force calculation. Contact conditions such as ***Contact_Automatic_Surface_to_Surface** is one of the most widely used conditions due to its ease of use. Since the definition of slave and master is not needed separately, it can be easily applied to the analysis of a model composed of many parts such as a car crash.

On the other hand, in the case of contact conditions such as * **Contact_Surface_to_Surface**, the normal direction of the shell or segment is important. Normal contact of parts colliding should be outward as shown in the picture below. In addition, the contact force calculation should include the shell thickness option (**SHLTHK**). The computational speed is that non-automatic contact conditions are more efficient. If the model is complex and consists of a large number of shell parts, automatic contact may be beneficial, even though it may take some time.



▲ Shell normal orientation in * Contact_Surface_to_Surface

• One-Way Contact

One-way contact is a condition that considers only one direction of contact and of course, the analysis time is shorter than a condition that considers both directions. The master and slave surfaces must be defined correctly or correct contact between the parts may not occur. Generally, the parts with large Rigid body and coarse Mesh are defined as Master and the parts with small and which are moving are defined as Slave. This is very effective for **Nodes-to-Surface** conditions, for example, when analyzing SPH particles or explosion analysis. The following contact conditions are applicable.

***Contact_Nodes_to_Surface**

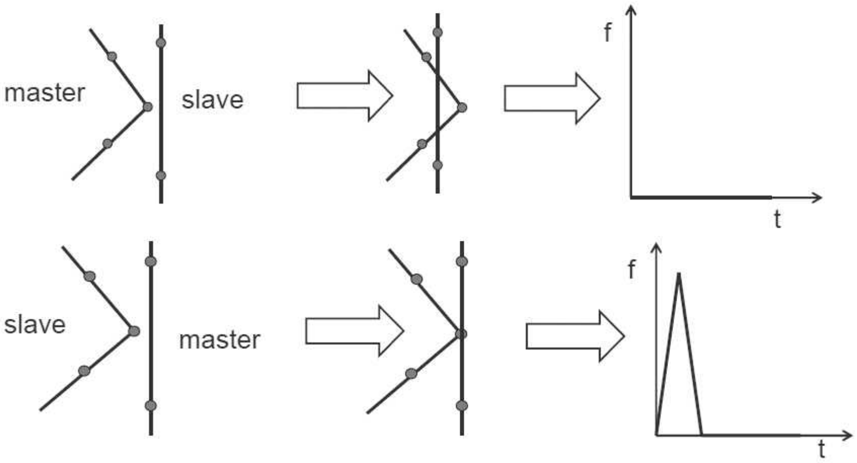
***Contact_One_way_Surface_to_Surface**

***Contact_Automatic_Nodes_to_Surface**

***Contact_Forming_Nodes_to_Surface**

***Contact_Eroding_Nodes_to_Surface**

***Contact_Constraint_Nodes_to_Surface**



▲ Contact condition at * Contact_One_way_Surface_to_Surface

• Single Surface Contact

Single surface contact applies when contact occurs within the same part, during buckling, or when contact occurs with the outer surface of another part. Only the Slave part is set, and the Master part does not need to be defined separately. Single surface contact is one of the easiest conditions to apply because LS-DYNA searches and defines the exterior surfaces of all components at the time of initialization. In the case of single surface contact, **RCFORC** result under contact is not outputted separately. Therefore, contact load calculation should be requested by using *** Contact_force_Tranducer_Penalty** keyword. **Automatic_Single _Surface** condition is one of the most widely used contact conditions in general collision analysis. In the case of a vehicle crash, a buckling phenomenon occurs, so a single surface contact condition should also be included. **Automatic_General** is also widely used, but it takes a relatively long time to solve. The types of Single Surface Contact are as follows.

***Contact_Single_Surface**

***Contact_Automatic_Single_Surface**

***Contact_Airbag_Single_Surface**

***Contact_General**

***Contact_Eroding_Single_Surface**

• Eroding contact

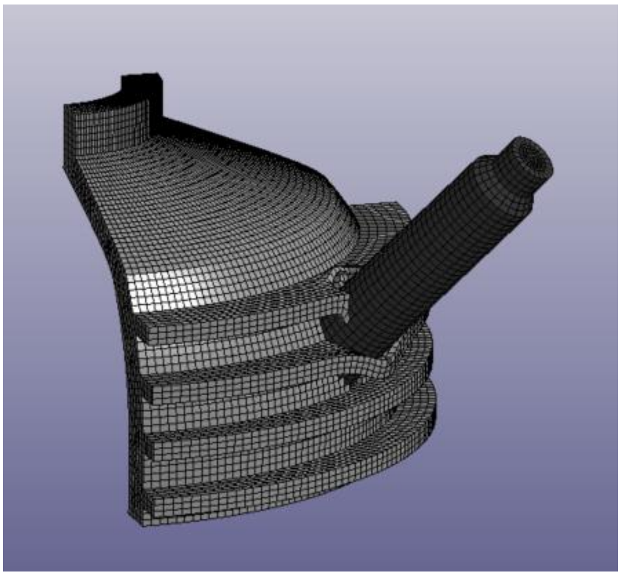
Eroding contact is a contact condition considering the case where a new contact surface is continuously generated due to element removal due to failure during the analysis process in the case of high-speed collision analysis using solid elements. This is the condition to be applied to the model where Penetration occurs as briefed in Example 9. That is, the analysis takes a long time because the contact surface must be updated continuously as the analysis proceeds, but it is a contact condition that can simulate the contact state most similar to the actual phenomenon.

The time step is continuously adjusted because the contact status is continuously checked and updated during the analysis process. These contact conditions include the following conditions, among which *** Contact_Eroding_Single_Surface** is most widely used for high-speed collision analysis such as ballistic impact.

***Contact_Eroding_Single_Surface**

***Contact_Eroding_Nodes_to_Surface**

***Contact_Eroding_Surface_to_Surface**



▲ *Contact_Eroding_Surface_to_Surface

• Tied contact

The contact condition for connecting and fixing the contact surfaces to each other is a Tied Contact condition. If the master surface is deformed during the analysis, the slave surface with the Tied contact condition is deformed together. In general, a part with a large mesh size can be designated as a master and tolerance can be entered even if the contact surfaces are separated from each other. It is good for connection between parts with different mesh size and it is a contact condition used when modeling the surface fixed by fasteners such as bolts or screws, or when bonding the interface with adhesive or glue material.

By inputting a failure criterion, the model can be modeled to fall under critical loads or stresses. For example, it can be simulated by setting tensile and shear loads as failure criteria using conditions such as * **Tiebreak_Nodes_to_Surface**. In the case of ***Tied_surface_to _surface_failure**, the contact condition can be simulated by setting the fracture stress as a reference. In general, when using * **Tied Contact**, rather than using the whole part, set the part as * **Set_Segment** to model only the local part.

A keyword that can simulate a similar phenomenon such as * **Tied_Contact** is * **Constrained_ <option>**. For example, keywords such as * **Contrained_spotweld** or ***Constrained_ generalized_Weld**, where certain parts are fixedly connected by welding and ***Constrained_ Nodal_Rigid_Body**, which constrain certain groups of nodes to move together, can be modeled similarly to tied contact conditions.

In particular, in the case of * **Constrained_NodaL_Rigid_Body**, in order to reduce the analysis time, this card can be applied by joining the part with the rigid body when stress or deformation of a particular part is not important. In the case of automobile collision analysis, the analysis time can be shortened by applying parts at the rear of the collision zone that is not particularly important in the collision analysis.

Tips for Contact Modeling

Here are some general tips for contact modeling. First of all, if you are not sure of the exact conditions during the initial modeling process, it is safe to apply ***Contact_Automatic _Single_Surface** and option **SOFT** = 1 to general explicit analysis first. This is the most effective and general method. First of all, the contact condition setting is simple. There is no need to define the Slave and Master part separately, especially in case of solving parallel analysis. One reason is that in parallel analysis, one macro contact condition is small and calculation can be performed faster than several separate contact conditions. The **SOFT** = 2 option may provide more accurate results, but it takes more computation time, so it is effective to prioritize **SOFT** = 0 or 1 in the model development process. Disadvantages are that the contact force generated at the contact surface cannot be calculated and there are not many options to choose from when the analysis time specifies master and slave individually. *** Contact_automatic_general** can be easily set, but it is best to avoid using it because it takes the longest analysis time.

Second, make sure that there are no overlapping elements between each other before performing the analysis. LS-DYNA finds the parts that are connected to each other during the initialization of the analysis model and automatically deletes them from the contact range. Overlapping can be checked through the **Keyword Manager → Model Check → Contact Check. If IGNORE = 2** is selected in the **Control** option, a report on the initial penetration is printed.

The key parameter of the contact load analysis is penetration stiffness ***'k'***, so check in advance that there will be no errors in the physical quantities that determine k (i.e. the material properties and shell thickness). In the case of non-automatic, in case of contact of shell element, make sure that shell normal direction is facing each other. If the contact between shell elements does not occur at all and penetrates, please check the orientation of shell elements first.

If one-way contact is applied, the master part should be selected where the part with higher rigidity or the part with larger overall size and relatively large mesh. The site where the contact occurs is as close as possible to produce the mesh size the most stable contact occurs. The more complex the contact condition, the smaller the contact damping parameter VDC is. It is recommended to use less than 10% (VDC = 10.0). Higher the damping value, the greater the instability, which can take longer to solve and interpret.

In addition, if the problem persists in contact results, it may be helpful to make the time step size smaller or the contact stiffness smaller. The time step is smaller than **TSSFAC** <0.9 in ***Control_Timestep** and **Contact_Stiffness** can be solved by reducing Modulus of the part where an impact occurs. One way is to reduce the mesh size of the impact area (fine mesh).

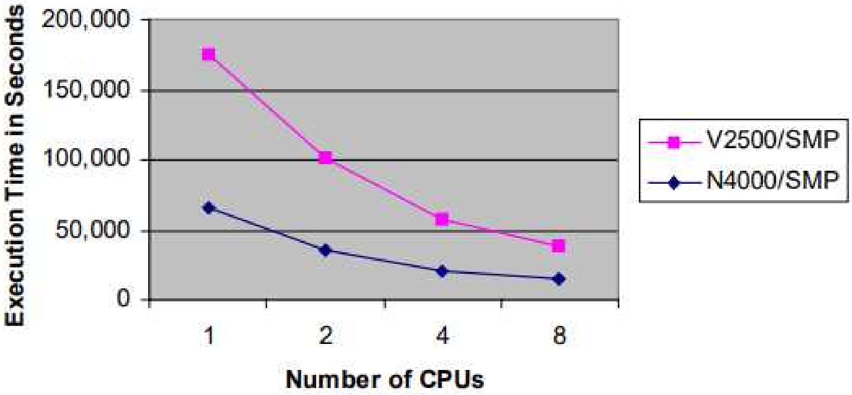
A4

A4. Parallel Analysis and Single/Double Precision

Efforts should be made to reduce the analysis time by using parallel analysis when analyzing collisions for large structures such as automobile collisions and aircraft and trains. LS-DYNA provides two functions, Shared Memory Parallel Processing (SMP) and Massively Parallel Processing (MPP). Individual readers may not have a choice, but most large companies have an MPP system and perform analysis.

SMP analysis means that the analysis model is distributed to multiple CPUs in one analysis machine and MPP means using all CPUs of multiple analysis machine clusters. In order to use SMP, Ncpu = No. of CPU must be entered. Although there are generally about 8 CPUs for personal computers in recent years and 32 CPUs for workstations, it is effective to use MPP when 5 or more CPUs are used by the calculation method. Up to 5 CPUs, the larger the number of CPUs, the more linearly the computation speed seems to decrease, but beyond the 5, the effect is negligible on SMP. In addition, it can be seen that the analysis results vary slightly each time since the method of distributing the calculation to the CPU when using SMP varies depending on the situation. Individual readers with fewer than five or six CPUs should, of course, use SMP. In the graph below, you can clearly see the difference in analysis time according to the number of CPUs.

.

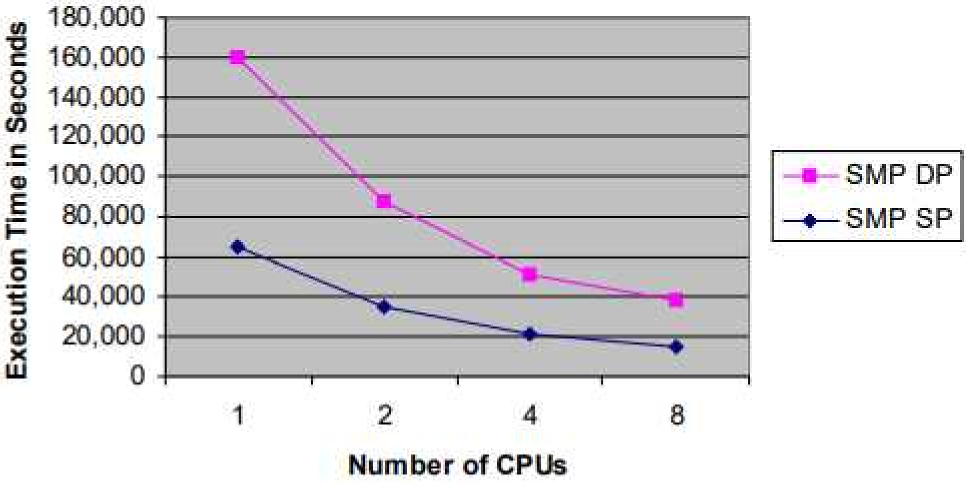


▲ Comparison of analysis time according to CPU number [11]

---

[11] Y.Y. Lin, C.D. Kan, A performance Study of LS-DYNA on Vehicle Crash Simulation, LS-DYNA International Conference, 2000

In addition, the analysis time may vary depending on what memory type is used, what is the precision method and what is the data communication method between the CPUs. If the option is available to the individual reader, SP (Single precision) and DP (Double precision) can be selected. When applying DP, it is known that the analysis time takes about 1.5 times longer than SP.



▲ Comparison of Analysis Times According to SP and DP [11]

Single Precision and Double Precision

Single Precision uses a 32-bit concept to represent numbers using 0-31 bits represented by 0,1. In the representation of numbers, the first bit is a sign, the next eight bits are exponents, and the remaining 23 bits are numbers.

```
S EEEEEEEE FFFFFFFFFFFFFFFFFFFFFFF
0 1      8 9                    31
```

▲ Single Precision Expression

```
0 00000000 00000000000000000000000 = 0
1 00000000 00000000000000000000000 = -0

0 11111111 00000000000000000000000 = Infinity
1 11111111 00000000000000000000000 = -Infinity

0 11111111 00000100000000000000000 = NaN
1 11111111 00100010001001010101010 = NaN

0 10000000 00000000000000000000000 = +1 * 2**(128-127) * 1.0 = 2
0 10000001 10100000000000000000000 = +1 * 2**(129-127) * 1.101 = 6.5
1 10000001 10100000000000000000000 = -1 * 2**(129-127) * 1.101 = -6.5
```

▲ Number expressed as Single Precision (example)

Double precision, on the other hand, is a representation that takes 64-bit units. The number is represented by 0-63 bits. In the representation of numbers, the first bit means sign, the next 11 bits are exponent and the remaining 52 bits are numbers.

```
S EEEEEEEEEEE FFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFF
0 1        11 12                                                63
```

▲ Double Precision expression

Thus, Precision can tell you how many decimal points of a number are valid. In the case of single-precision, it is valid to 7 ~ 8 digits because it is Log_{10} (2^{24}). In the case of double-precision, it is safe to use 15 ~ 16 digits because it is $\log_{10}$ (2^{53}). It may be a bit difficult to understand, but I think that Single-Precision is effective for the interpretation of individual users, not for the interpretation of Science Research. Of course, double precision always gives more accurate results than Single. Naturally, the analysis takes a bit longer and the effect of the difference in the analysis will not be greater than factors such as the size, boundary conditions, and load conditions of the elements in other models. If you ask a nearby engineer this concept, they might not be aware of this. However, if you are an engineer whose primary focus is analysis, you should be familiar with this concept.

A5

A5. Convergence Problems

The convergence in FEM analysis is a problem that can occur in all analysis fields, not just the case of using LS-DYNA's Implicit. This is a common problem in the nonlinear analysis if the static equilibrium is not maintained. In particular, models that contain a lot of contacts, models containing high nonlinear materials such as rubber or tires and models that include advanced failure theory will not be easy to solve problems due to convergence. In this chapter, I'll give you tips on how to find and solve problems with convergence. As the analysis progresses, the problem of convergence is affected by the following factors:

- **Mesh Size:** In general, large elements do not look good but do not converge well.
- **Time / Load step size:** If the step size to apply the load is too large, it will not converge.
- **Rigid body motion:** Load and boundary conditions must be established to maintain equilibrium.
- **Contact:** Initial penetration, excessive contact load, contact displacement, etc., those impede convergence.
- **Material property:** High nonlinearity has a big impact on convergence.

What's Convergence

Basically, FEM produces the result by solving the Matrix Equation. If it is a simple linear model, it is the process of solving the following equation and if it is nonlinear, it is the process of solving the equation including one more term.

- **Linear Model** $[K]\{u\} = \{f\}$
- **Non-linear model** $[K(u)]\{u\} = \{f\}$

In the linear model, the matrix ***[K]*** is constant and independent of the variable ***{u}***. Thus, while it is easy to find a variable, in a nonlinear analysis it is not possible to compute the ***{u}*** matrix until the ***[K(u)]*** variable is found.

- When starting the FEM calculation, first calculate the initial displacement.

$$\{u\} = \{u^0\} = 0$$

- Then, ***$\{u^0\}$*** is known, we can compute the ***$[K(u^0)]$*** matrix.

- Once the matrix has been calculated, we can solve the governing equations of the system.

$$[K(u^0)]\{u^1\} = \{f\}$$

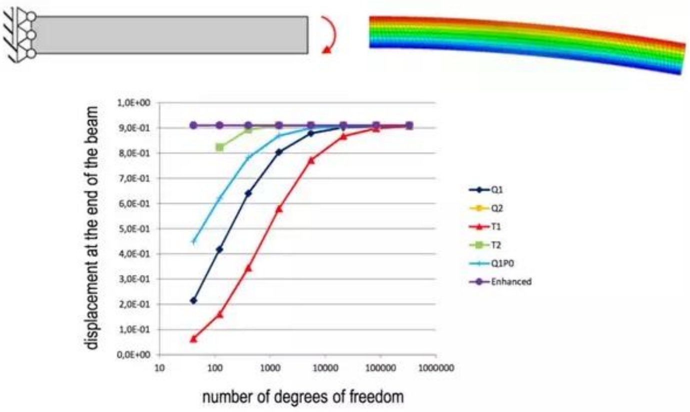
- However, because FEM is not an exact solution but an approximate solution, the following equation does not always hold.

$$[K(u^1)]\{u^1\} - \{f\} \neq 0$$

- Therefore, it is necessary to find ***$\{u^n\}$*** through the iteration so that the following value is as close to zero as possible.

$$[K(u^n)]\{u^n\} - \{f\} \approx 0$$

Numerically, it can never reach zero. There is a cut-off value set for each program and it can be estimated that convergence was reached when this range was reached. In LS-DYNA, it can be set in **DCTOL** (Displacement Tolerance) of *** Control_Implicit_Solution** keyword. The most common issue with convergence is the problem of the mesh size. As the size of the element decreases, the result changes within a certain range as the result changes, so how small can you evaluate the mesh size to obtain reliable results?



▲ A convergence of Analysis Results According to Mesh Size

Reducing the mesh size to steadily increasing stresses or continually increasing displacements is an easy experience for everyone. In general, even when the size of the mesh is 1/2 if the result is less than 5% of the differences in stress level, then, the mesh convergence can be accomplished. This is not a law set by anyone, but an engineering judgment commonly used in the engineering field.

The second problem is the **Unstable System**. The unstable system represented that the FEM model is unrealistic compared to an actual target system. If the force continues to push in one direction but the boundary condition to support the force which is not set, the rigid body motion occurs, or the stress and displacement occur when the contact occurs. This is the situation where a static equilibrium cannot be achieved, consequently, convergence could not be maintained. This is the part that needs to be solved by looking at the model macroscopically. In addition, convergence problems occur when too large step sizes are applied outside the convergence region or when the deformation of the material with nonlinear properties does not occur within the convergence region. Let's see how to solve these problems concretely in LS-DYNA.

LS-DYNA Convergence Problem Solution

Convergence problems that occur when using the implicit solver in DYNA can be solved in several ways.

- **Check the mesh quality in the model.**

If the quality of the mesh created through **automesh** is poor, convergence may occur. Use the **Mesh quality check** feature in Model Check to improve the shape of a mesh that is of poor quality (ie, dents on one side of 2D elements, or poor aspect ratio of solid elements can cause negative volume errors). Also, check the free edge to see if there are any meshes that are not properly connected. In particular, pay close attention to the area where the beam and shell and the shell and solid are connected.

- **Check the unit of the model.**

If the unit of the model is not applied consistently, it is very likely to cause a numerical error, so the unit of the model and material property should be checked. In particular, the properties of heat, properties of energy, and properties of the fluid can easily fall into error.

- **Verify that there is no initial penetration in the contact.**

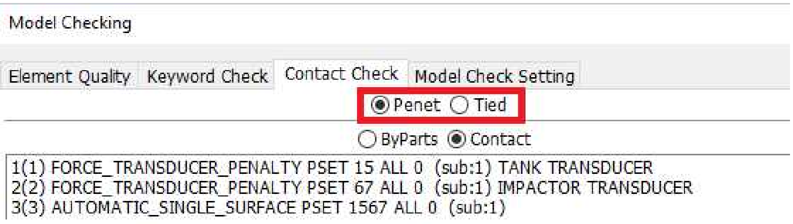
Models with multiple parts are more likely to take for consideration while solving, so check them using the **Model check -Penetration check** function. If the modification of the penetration site is difficult, the **IGNORE** = 1 of the **Control** keyword can be set to exclude the corresponding site from the contact solving. The more complicated and multiple the contact area, the smaller the step size should be created.

- **If you are using solid elements, use ELFORM = 1, -1, 13, 16.**

Also, if you are using shell elements, ELFORM = 6, 16, 16 can help with convergence. These elements use Under (Reduced) integration, which can be really helpful for convergence even with poor mesh quality.

- **Check if the Tied Contact is properly connected.**

If the slave node is not connected to the master node as intended, the rigid body behavior may occur, causing problems with convergence. Slave nodes should be smaller than the master, but if there are other parts, the corresponding parts should be checked carefully. There is a Tied contact check function in Model checking and prior to solving the analysis, this can check which nodes are Tied and also it's written in the D3hsp file.



● Penet ○ Tied

○ ByParts ● Contact

```
1(1) FORCE_TRANSDUCER_PENALTY PSET 15 ALL 0  (sub:1) TANK TRANSDUCER
2(2) FORCE_TRANSDUCER_PENALTY PSET 67 ALL 0  (sub:1) IMPACTOR TRANSDUCER
3(3) AUTOMATIC_SINGLE_SURFACE PSET 1567 ALL 0  (sub:1)
```

▲ *Tied contact check

- **Set the Viscous Damping option VDC = 0.0.**

If damping is set to a large value, implicit solver acts as a deterrent to convergence.

- **Set the Initial time step size smaller.**

How small should it be? First of all, let's analyze by the default step size. If an error that doesn't converge occurs, reduce it to 50% of the existing value and re-analyze it. There is also an option to automatically adjust the time step via **Control_Implicit_Auto**. It may also be helpful to increase the **DCTOL** in the * **Control_Implicit_Solution** keyword or to increase the **MAXREF** (maximum number of operations) to 30-60.

- **Check for rigid body motion.**

In most cases, rigid body motion occurs when components are not connected or if the boundary condition is set incorrectly. In this case, the message **'Negative eigenvalue warning'** is displayed on the log screen. Looking at the **D3hsp** file, you can see in more detail what part is occurring. If you check the text output file **'EIGOUT',** for the first 10 Eigenvalues, including the * **Control_Implicit_Eigenvalue** keyword in the model, the model where Rigid body motion occurs will be almost given zero EIGOUT value. In this case, there is a problem in the contact condition or boundary condition, so check those conditions.

- **Switch from Load control to Displacement control.**

In some cases of unstable systems where severe buckling or sudden displacements occur in the model, the application of displacement instead of load improves convergence. Using ***Control_Implcit_Dynamics** also helps with convergence.

- **If you open the D3iter file (Binary), you can visually check the Convergence problem area.**

If the analysis is interrupted due to convergence problems, let's open the **D3iter** file and observe the shape of the deformation up to the point where the analysis proceeded. It can be seen that a problem occurs such that a particular part is blown up by an improper constraint or a sudden change of a specific part occurs. Use **RESPLT** = 1 in the ***Database_Extent_Binary** keyword to identify the problem area more accurately.

There are many other methods, but the points described so far will be fair enough to identify and solve the problem. The most important thing is the logical thinking of the interpreter and the inclination to solve the problem and the accumulated experience based on this.

A6

A6. Useful Reference Site

We have compiled a list of sites that you can refer to for your further reference. LS-DYNA is a program with a very active user group worldwide. In addition, a lot of technical data and articles using LS-DYNA are published in many journals. There are many useful materials and analysis models available on the internet for learning purposes. So, you will be able to learn your own specific analysis problem using available technical materials.

www.lsdynasupport.com	Provide various learning materials
www.dynasupport.com	Provide various learning materials
www.DYNAExamples.com	Many example models
www.dynalook.com	LS-DYNA Conference Paper
www.FEAInformation.com	Newsletter site
www.DYNAmore.com	Seminar information
www.D3View.com	Tip blog
LS-DYNA@yahoogroups.com	Yahoo user group
Awg.lstc.com	Aerospace field LS-DYNA technical data
www.nhtsa.gov/crash-simulation-vehicle-models	Vehicle crash models available

Intentional Blank

EPILOGUE

This book was written for beginning engineers and graduate students who have no experience with LS-DYNA. In the future, I will prepare more professional books that contains a lot of knowledge and know-how. The author would like to thank the readers who study the subject of the FEM. Also, the author would like to express sincere appreciation to Puneeth Latha for his helpful comments on this edition. Hopefully, this book will improve the FEM problem solving ability of many readers using LS-DYNA. Please let me know if you need any further help. Also, please contact us if you would like to work on the project together.

October, 2019
Ryan Lee